CNC 8055 M

Operating Manual

Ref. 9909 (in)

Please note that the availability of some of the features described in this manual depends on the software options you just obtained.

MODEL	GP	M
Electronic threading	Not available	Available
Tool magazine management	Not available	Available
Solid Graphics	Not available	Option
Machining canned cycles	Not available	Available
Multiple machining	Not available	Available
Probing canned cycles	Not available	Option
Tool life monitoring	Not available	Option
Irregular pockets with islands	Not available	Option
Digitizing	Not available	Option
Tracing	Not available	Option
TCP transformation	Not available	Option
Tool radius compensation	Option	Available
DNC	Option	Option
Software for 7 axes	Option	Option
Profile editor	Option	Option
Rigid tapping	Option	Option
Tangential control	Not available	Option
Conversational Software (MC model)	Not available	Option

---------- o ----------

The information described in this manual may be subject to variations due to technical modifications.

FAGOR AUTOMATION, S.Coop. Ltda. reserves the right to modify the contents of the manual without prior notice.

INDEX

Section Page

VERSION HISTORY

INTRODUCTION

Safety conditions 3
Material returning terms 5
Fagor documentation for the CNC 6
Manual contents 7

1. OVERVIEW

1.1 Part-programs 1
1.2 Monitor information layout 4
1.3 Keyboard layout 6
1.4 Operator panel layout 8

2 OPERATING MODES

2.1 Help systems 3
2.2 Software update 5

3 EXECUTE / SIMULATE

3.1 Block selection and stop condition 4
3.2 Display selection 7
3.2.1 Standard display mode 9
3.2.2 Position display mode 10
3.2.3. Part program display mode 10
3.2.4. Subroutine display mode 11
3.2.5 Following error display mode 14
3.2.6 User display mode 14
3.2.7 Execution time display mode 15
3.3 Mdi 17
3.4 Tool inspection 18
3.5 Graphics 20
3.5.1 Type of graphics 21
3.5.2 Display area 26
3.5.3 Zoom 27
3.5.4 Viewpoint 28
3.5.5 Graphic parameters 29
3.5.6 Clear screen 31
3.5.7 Deactivate graphics 31
3.5.8 Measure 32
3.6 Single block 33

Section Page

4. EDIT

4.1 Edit 2
4.1.1 Editing in cnc language 2
4.1.2 Teach-in editing 3
4.1.3 Interactive editor 4
4.1.4 Profile editor 5
4.1.4.1 Operation with the profile editor 6
4.1.4.2 Profile editing 7
4.1.4.3 Definition of a straight section 8
4.1.4.4 Definition of a circular section 9
4.1.4.5 Corners 10
4.1.4.6 Modify 11
4.1.4.7 Finish 13
4.1.4.8 Examples of profile definition 14
4.2 Modify 18
4.3 Find 19
4.4 Replace 20
4.5 Delete block 21
4.6 Move block 22
4.7 Copy block 23
4.8 Copy to program 24
4.9 Include program 25
4.10 Editor parameters 26
4.10.1 Autonumbering 26
4.10.2 Axes selection for teach-in editing 27

5. JOG

5.1 Jogging the axes 9
5.1.1 Continuous jog 9
5.1.2 Incremental jog 10
5.1.3 Jogging with electronic handwheel 11
5.1.3.1 The machine has one electronic handwheel 11
5.1.3.2 The machine has several handwheels 12
5.2. Manual control of the spindle 13

6. TABLES

6.1 Zero offset table 2
6.2 Tool offset table 3
6.3 Tool table 4
6.4 Tool magazine table 6
6.5 Global and local parameter tables 7
6.6 How to edit tables 8

Section Page

7. UTILITIES

7.1 Directory 1
7.1.1 Directory of the external devices 3
7.2 Copy 4
7.3 Delete 4
7.4 Rename 5
7.5 Protections 6
7.6 Change date 7

8. STATUS

8.1 CNC 1
8.2 DNC 2

9. PLC

9.1 Edit 2
9.2 Compile 9
9.3 Monitoring 10
9.3.1 Monitoring with the plc in operation and with the plc stopped 17
9.4 Active messages 19
9.5 Active pages (screens) 19
9.6 Save program 19
9.7 Restore program 20
9.8 Resources in use 20
9.9 Statistics 21
9.10 Logic analyzer 23
9.10.1 Description of the work screen 23
9.10.2 Selection of variables and trigger conditions 26
9.10.2.1 Variable selection 26
9.10.2.2 Selection of trigger condition 28
9.10.2.3 Selection of time base 30
9.10.3 Execute trace 31
9.10.3.1 Data capture 32
9.10.3.2 Modes of operation 33
9.10.3.3 Trace representation 34
9.10.4 Analyze trace 35

10. SCREEN EDITOR

10.1 Utilities 3
10.2 Editing custom screens (pages) and symbols 5
10.3 Graphic elements 10
10.4 Texts 15
10.5 Modifications 18

Section Page

11. MACHINE PARAMETERS

11.1 Machine parameter tables 2
11.2 Miscellaneous function tables 3
11.3 Leadscrew error compensation tables 4
11.4 Cross compensation tables 5
11.5 Operation with parameter tables 6

12. DIAGNOSIS

12.1 Configuration 2
12.1.1 Hardware configuration 2
12.1.2 Software configuration 3
12.2 Hardware test 4
12.3 Memory test 5
12.4 Flash memory test 5
12.5 User 5
12.6 Hard disk 5
12.7 Interesting notes 6

VERSION HISTORY (M)

(MILL MODEL)

Date: **May 1999** ***Software Version:*** **3.0x**

FEATURE	AFFECTED MANUAL & CHAPTERS	
Portuguese language	Installation Manual	Chapter 3
Tangential Control	Installation Manual Programming Manual	Chapters 9, 10, Appendix Chapters 6, 13, Appendix
Incline planes. The software travel limits are monitored in JOG movements.		
PLC. User registers R1 through R499	Installation Manual Programming Manual	Chapters 6, 7, Appendix Chapter 13
CNC status screen	Operation Manual	Chapter 8
Hard disk (HD)	Installation Manual	Chapters 1, 3, Appendix
HD Diagnosis	Operation Manual	Chapter 12
Integrate the HD into an outside PC network	Installation Manual	Chapter 3
Consult directories, delete, rename and copy programs in the same or other device	Operation Manual Programming Manual	Chapters 1, 7 Chapter 1
Ejecution and simulacion from RAM memory, Memkey Card, HD or serial line.	Operation Manual	Chapters 1, 3,
It is possible to execute (EXEC) and open (OPEN) a program (to be edited) stored in any device.	Programming Manual	Chapter 14, Appendix
MC option. Tool calibration screen. When defining R and L; I and K are initialized If I=0 and K=0; I and K are initialized	Operation Manual	Chapter 3
MC option. ISO management, also as MDI	MC Operation Manual	Chapter 3
MC option. New way to handle safety planes.	MC Operation Manual	Chapter 4
MC option. New codes for specific keys.	MC Operation Manual	Appendix

INTRODUCTION

SAFETY CONDITIONS

Read the following safety measures in order to prevent damage to personnel, to this product and to those products connected to it.

This unit must only be repaired by personnel authorized by Fagor Automation.

Fagor Automation shall not be held responsible for any physical or material damage derived from the violation of these basic safety regulations.

Precautions against personal damage

Before powering the unit up, make sure that it is connected to ground

In order to avoid electrical discharges, make sure that all the grounding connections are properly made.

Do not work in humid environments

In order to avoid electrical discharges, always work under 90% of relative humidity (non-condensing) and 45º C (113º F).

Do not work in explosive environments

In order to avoid risks, damage, do no work in explosive environments.

Precautions against product damage

Working environment

This unit is ready to be used in Industrial Environments complying with the directives and regulations effective in the European Community

Fagor Automation shall not be held responsible for any damage suffered or caused when installed in other environments (residential or homes).

Install the unit in the right place

It is recommended, whenever possible, to instal the CNC away from coolants, chemical product, blows, etc. that could damage it.

This unit complies with the European directives on electromagnetic compatibility. Nevertheless, it is recommended to keep it away from sources of electromagnetic disturbance such as.

- Powerful loads connected to the same AC power line as this equipment.
- Nearby portable transmitters (Radio-telephones, Ham radio transmitters).
- Nearby radio / TC transmitters.
- Nearby arc welding machines
- Nearby High Voltage power lines
- Etc.

Ambient conditions

The working temperature must be between +5° C and +45° C (41ºF and 113º F)
The storage temperature must be between -25° C and 70° C. (-13º F and 158º F)

Protections of the unit itself

Power Supply Module

It carries two fast fuses of 3.15 Amp./ 250V. to protect the mains AC input

Axes module

All the digital inputs and outputs have galvanic isolation via optocouplers between the CNC circuitry and the outside.
They are protected by an external fast fuse (F) of 3.15 Amp./ 250V. against reverse connection of the power supply.

Input / Output Module

All the digital inputs and outputs have galvanic isolation via optocouplers between the CNC circuitry and the outside.
They are protected by an external fast fuse (F) of 3.15 Amp./ 250V. against a voltage overload (greater than 33Vdc) and against reverse connection of the power supply.

Input / Output and Tracing Module

All the digital inputs and outputs have galvanic isolation via optocouplers between the CNC circuitry and the outside.
They are protected by an external fast fuse (F) of 3.15 Amp./ 250V. against a voltage overload (greater than 33Vdc) and against reverse connection of the power supply.

Fan Module

It carries 1 or 2 external fuses depending on model
The fuses are fast (F), of 0.4 Amp./ 250V. to protect the fans.

Monitor

The type of protection fuse depends on the type of monitor. See the identification label of the unit itself.

Precautions during repair

Do not manipulate the inside of the unit

Only personnel authorized by Fagor Automation may manipulate the inside of this unit.

Do not manipulate the connectors with the unit connected to AC power.

Before manipulating the connectors (inputs/outputs, feedback, etc.) make sure that the unit is not connected to AC power.

Safety symbols

Symbols which may appear on the manual

WARNING. symbol
It has an associated text indicating those actions or operations may hurt people or damage products.

Symbols that may be carried on the product

WARNING. symbol
It has an associated text indicating those actions or operations may hurt people or damage products.

"Electrical Shock" symbol
It indicates that point may be under electrical voltage

"Ground Protection" symbol
It indicates that point must be connected to the main ground point of the machine as protection for people and units.

MATERIAL RETURNING TERMS

When returning the Monitor or the Central Unit, pack it in its original package and with its original packaging material. If not available, pack it as follows:

1.- Get a cardboard box whose three inside dimensions are at least 15 cm (6 inches) larger than those of the unit. The cardboard being used to make the box must have a resistance of 170 Kg (375 lb.).

2.- When sending it to a Fagor Automation office for repair, attach a label indicating the owner of the unit, person to contact, type of unit, serial number, symptom and a brief description of the problem.

3.- Wrap the unit in a polyethylene roll or similar material to protect it.

When sending the monitor, especially protect the CRT glass

4.- Pad the unit inside the cardboard box with poly-utherane foam on all sides.

5.- Seal the cardboard box with packing tape or industrial staples.

FAGOR DOCUMENTATION FOR THE CNC

OEM Manual	Is directed to the machine builder or person in charge of installing and starting-up the CNC.
USER Manual	Is directed to the end user or CNC operator. It contains 2 manuals: Operating Manual describing how to operate the CNC. Programming Manual describing how to program the CNC.
DNC Software Manual	Is directed to people using the optional DNC communications software.
DNC Protocol Manual	Is directed to people wishing to design their own DNC communications software to communicate with the CNC.
FLOPPY DISK Manual	Is directed to people using the Fagor Floppy Disk Unit and it shows how to use it.

MANUAL CONTENTS

The operating Manual for the Mill model CNC contains the following chapters:

Index

New features and modifications for the Mill Model.

Introduction
Summary of safety conditions
Shipping terms
Fagor documentation for the CNC
Manual contents.

Chapter 1 Overview
Location of the part-programs, how to edit and execute them.
It indicates the layout of the keyboard, operator panel and of the data on the monitor.

Chapter 2 Operating modes.
Description of the different operating modes of the CNC.

Chapter 3 Execute / Simulate
It describes how to operate in the "Execution" and "Simulation" modes.
Both operations may be performed in automatic or single block mode.

Chapter 4 Edit
Description of the "Edit" mode of operation.
The different ways to edit a part-program are: in CNC language, in Teach-in mode, using the Interactive editor and the Profile editor.

Chapter 5 Jog
Description of the "Jog" mode of operation.
This is the operating mode to be used whenever the machine is to be controlled manually to move the axes of the machine as well as to control the spindle.

Chapter 6 Tables
Description of the "Tables" mode of operation.
It allows access to the various data tables of the CNC: Zero offsets, Tool offsets, Tool table, tool magazine and global and local arithmetic parameters.

Chapter 7 Utilities
Description of the "Utilities" mode of operation.
It allows access to the directory of part-programs, subroutines and to the part-program directory of the PC or peripheral device connected to the CNC. It is also possible to copy, delete, move or rename part-programs.
It indicates the protections that could be assigned to a part-program.
It shows the various ways to operate with the Flash memory.

Chapter 8 Status
It shows the status of the "CNC" and DNC communication lines.
It describes the "DNC" mode of operation and how to operate via serial interfaces.

Chapter 9 PLC
Description of the "PLC" mode of operation.
It shows how to edit and compile the PLC program
It is possible to verify how the PLC program works and the status of its numerous variables.
It shows the date the PLC program was edited, its memory size and the execution times (cycle times) for its different modules.
It offers a detailed description of the logic analyzer.

Chapter 10 Graphic Editor

Description of the "Graphic Editor" mode of operation".
It indicates how to create user defined pages (screens) and symbols to create user screens.
It shows how to use user pages in customizing programs, how to display a user page on power-up and how to activate user pages from the PLC.

Chapter 11 Machine parameters

Description of the "Machine parameters" mode.
It is possible to access and operate with the tables for machine parameters, miscellaneous "M" functions, leadscrew error compensation and cross compensation.

Chapter 12 Diagnosis

Description of the "Diagnosis" mode
It is possible to know the CNC configuration and run a system test.

1. OVERVIEW

In this manual an explanation is given of how to operate the CNC by means of its Monitor-Keyboard unit and the Operator Panel.

The Monitor-Keyboard unit consists of:

* The Monitor or CRT screen, which is used to show the required system information.

* The Keyboard, which allows communication with the CNC, allowing information to be requested by means of commands or by changing the CNC status by generating new instructions.

1.1 PART-PROGRAMS

Editing

To create a part-program, access the Edit mode. See chapter 5 in this manual.

The new part-program edited is stored in the CNC's RAM memory.

A copy of the part-programs may be stored in the "MemKey Card", at a PC connected through serial line 1 or 2 or in the hard disk (HD module). See chapter 7 in this manual.

When using a PC through serial line 1 or 2, proceed as follows:
- Execute the "Fagor50.exe" applications program at the PC.
- Activate DNC communications at the CNC. See chapter 8 in this manual.
- Select the work directory as shown in chapter 7 of this manual. Option: Utilities\ Directory\ Serial L.\ Change directory.

With the Edit mode of operation, part-programs residing in the CNC's RAM memory may be modified. To modify a program stored in the "MemKey Card", in a PC or in the hard disk, it must be previously copied into RAM memory.

Execution

Part-programs stored anywhere may be executed or simulated. See chapter 3 in this manual.

The user customizing programs must be in RAM memory so the CNC can execute them.

The GOTO and RPT instructions cannot be used in programs executed from a PC connected through the serial lines. See chapter 14 of the programming manual.

The subroutines can only be executed if they reside in the CNC's RAM memory. Therefore, to execute a subroutine stored in the "MemKey Card", in a PC or in the hard disk, it must be first copied into the CNC's RAM memory.

From a program in execution, another program can be executed which is in RAM memory, in the "MemKey Card", in a PC or in the hard disk using the EXEC instruction. See chapter 14 of the programming manual.

Utilities

This operating mode, chapter 7 of this manual, lets display the part-program directory of all the devices, make copies, delete, rename and even set the protections for any of them.

Ethernet

When having the Ethernet option and if the CNC is configured as another node within the computer network, the following operations are possible from any PC of the network:

- Access the part-program directory of the Hard Disk(HD).
- Edit, modify, delete, rename, etc.the programs stored on the hard disk (HD).
- Copy programs from the hard disk to the PC and vice versa.

To configure the CNC as another node within the computer network, see section 3.3.4 of the installation manual.

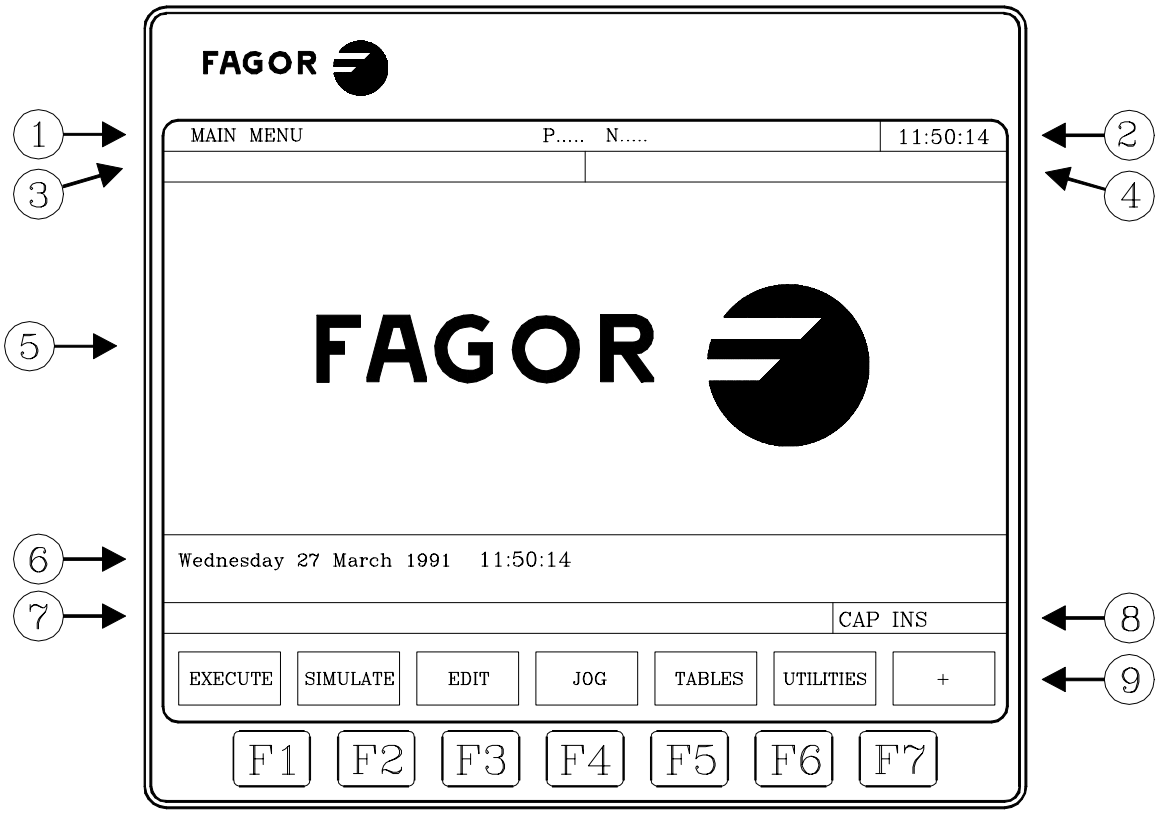
Operations that may be carried out with part-programs:

	RAM Memory	CARD A	HD	DNC
Consult the program directory in ...	Yes	Yes	Yes	Yes
Consult the subroutine directory in ...	Yes	No	No	No
Create work directory in ..	No	No	No	No
Change work directory in ..	No	No	No	Yes
Edit a program in ..	Yes	No	No	No
Modify a program in ..	Yes	No	No	No
Delete a program from ..	Yes	Yes	Yes	Yes
Copy from/to RAM memory to/from ...	Yes	Yes	Yes	Yes
Copy from/to CARD A to/from ...	Yes	Yes	Yes	Yes
Copy from/to HD to/from ...	Yes	Yes	Yes	Yes
Copy from/to DNC to/from ...	Yes	Yes	Yes	Yes
Rename a program in ..	Yes	Yes	Yes	No
Change the comment of a program in ..	Yes	Yes	Yes	No
Change protections of a program in ..	Yes	Yes	Yes	No
Execute a part-program in ..	Yes	Yes	Yes	Yes
Execute a user program in ..	Yes	No	No	No
Execute the PLC program in ..	Yes	*	No	No
Execute programs using the GOTO or RPT instructions from ..	Yes	Yes	Yes	No
Execute subroutines stored in ..	Yes	No	No	No
Execute programs stored in RAM, CARD A or HD using the EXEC instruction from ..	Yes	Yes	Yes	Yes
Execute programs via DNC with the EXEC instruction from ..	Yes	Yes	Yes	No
Open programs stored in RAM, CARD A or HD using the OPEN instruction from ..	Yes	Yes	Yes	Yes
Open programs via DNC using the OPEN instruction from ..	Yes	Yes	Yes	No
Consult from a PC and through Ethernet, the program directory in ...	No	No	Yes	No
Consult from a PC and through Ethernet, the subroutine directory in ...	No	No	No	No
Create from a PC and through Ethernet, a directory in...	No	No	No	No

* If it is not in RAM memory, it generates an executable code in RAM and it executes it..

1.2 MONITOR INFORMATION LAYOUT

The monitor is divided into the following areas or display windows:



1.- This window indicates the selected operating mode, as well as the program number and the number of active blocks.

The program status is also indicated (in execution or interrupted) and if the DNC is active.

2.- This window indicates the time in the " hours : minutes : seconds " format.

3.- This window displays the Messages sent to the operator from the part program or via DNC.

The last message received will be shown regardless of where it has come from.

4.- This window will display messages from the PLC.

If the PLC activates two or more messages, the CNC will always display the one with the highest priority, which is the message with the smallest number. In this way, MSG1 will have the highest priority and MSG128 will have the lowest.

In this case the CNC will display the character + (plus sign), indicating that there are more messages activated by the PLC, it being possible to display them if the ACTIVE

MESSAGE option is accessed in the PLC mode.

In this window the CNC will also display the character ***** (asterisk), to indicate that at least one of the 256 user-defined screens is active.

The screens which are active will be displayed, one by one, if the ACTIVE PAGES option is accessed in the PLC mode.

5.- Main window.

Depending on the operating mode, the CNC will show in this window all the information necessary.

When a CNC or PLC error is produced the system displays this in a superimposed horizontal window.

The CNC will always display the most important error and it will show:

* The "down arrow" key to indicate that another less important error has also occurred and to press this key to view its message.

* The "up arrow" key to indicate that another more important error has also occurred and to press this key to view its message.

6.- Editing window.

In some operating modes the last four lines of the main window are used as editing area.

7.-CNC communications window (errors detected in edition, nonexistent program, etc.)

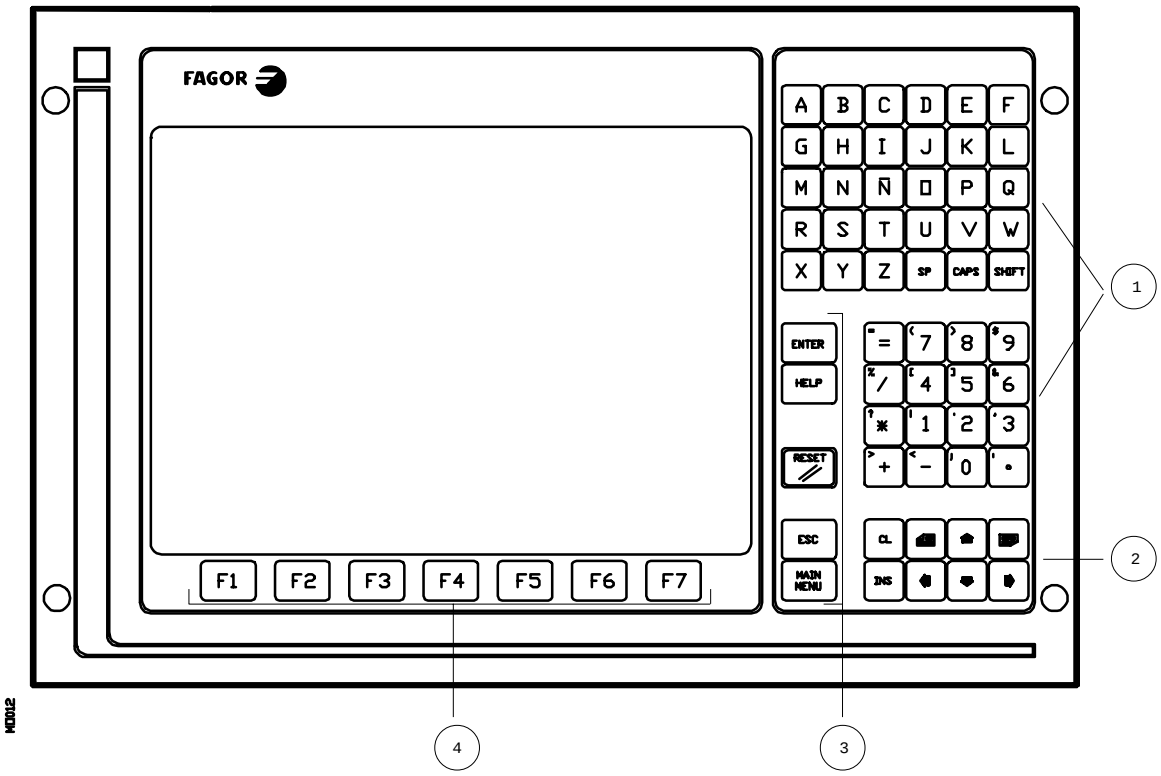
8.- This window displays the following information:

SHF	Indicates that the SHIFT key has been pressed to activate the second function of the keys. For example, if key is pressed after the SHIFT key, the CNC will understand that the "$" character is required.
CAP	This indicates capital letters (CAPS key). The CNC will understand that capital letters are required whenever this is active.
INS/REP	Indicates if it is insert mode (INS) or substitution (REP) mode. It is selected by means of the INS key.
MM/INCH	Indicates the unit system (millimeters or inches) selected for display.

9.- Shows the different options which can be selected with soft-keys F1 thru F7.

1.3 KEYBOARD LAYOUT

In accordance with the use of the different keys, it can be understood that the CNC keyboard is divided in the following way:



1.- Alphanumeric keyboard for the data entry in memory, selection of axes, tool offset, etc.

2.- Keys which allow the information shown on screen to be moved forward or backward, page to page or line to line, as well as moving the cursor all over the screen.

The CL key allows the character over which the cursor is positioned or the last one introduced, if the cursor is at the end of the line, to be erased.

The INS key allows the insert or substitution mode to be selected.

3.- Group of keys which due to their characteristics and importance are detailed below:

ENTER Used to validate CNC and PLC commands generated in the edition Window.

HELP Allows access to the help system in any operating mode.

RESET Used for initializing the history of the program in execution, by assigning it the values defined by machine parameters. It is necessary for the program to be stopped for the CNC to accept this key.

ESC Allows going back to the previous operating option shown on the monitor.

MAIN MENU When this key is pressed we can access the main CNC menu directly.

4.- SOFTKEYS or function keys which allow different operating options to be selected and which are shown on the monitor.

In addition, there are the following special keyboard sequences:

SHIFT RESET The result of this keystroke sequence is the same as if the CNC is turned off and turned back on. This option must be used after modifying the machine parameters of the CNC for these to be effective.

SHIFT CL With this keystroke sequence the display on the CRT screen disappears. To restore the normal state just press any key.

If, when the screen is off, an error is produced or a message from the PLC or CNC is received, the normal status of the screen will be restored.

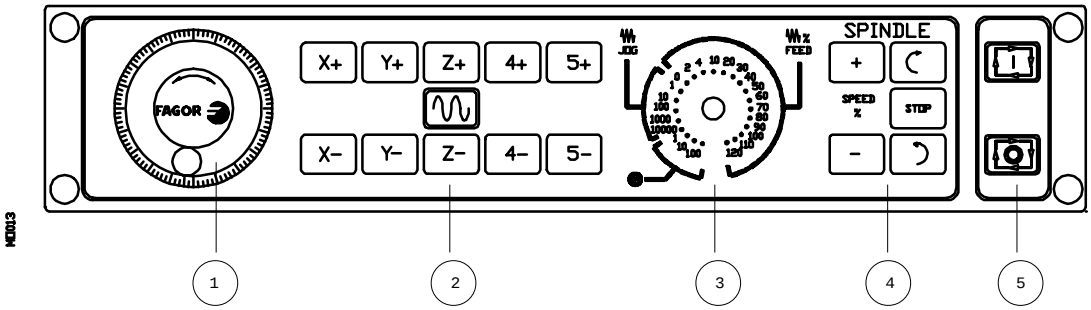
SHIFT This allows the position of the axes to be displayed on the right hand side of the screen as well as the status of the program being executed.

This can be used in any operating mode.

In order to recover the previous display it is necessary to press the keys using the same sequence.

1.4 OPERATOR PANEL LAYOUT

According to the utility which the different parts have, it can be considered that the Operator Panel of the CNC is divided in the following way:



1.- Position of the emergency button or electronic handwheel.

2.- Keyboard for manual movement of axes.

3.- Selector switch with the following functions:

Select the multiplication factor of the number of pulses from the electronic handwheel (1, 10 or 100).

Select the incremental value of the movement of the axes in movements made in the "JOG" mode.

Modify the programmed axis feedrate between 0% and 120%

4.- Keyboard which allows the spindle to be controlled, it being possible to activate it in the desired direction, stop it or vary the programmed turning speed between percentage values established by means of spindle machine parameters "MINSOVR" and "MAXOVR", with an incremental step established by means of the spindle machine parameter "SOVRSTEP".

5.- Keyboard for CYCLE START and CYCLE STOP of the block or program to be executed.

2. *OPERATING MODES*

After turning on the CNC, or after pressing the sequence of SHIFT-RESET keys, the FAGOR logo will appear in the main window of the monitor or the screen previously prepared as page 0 by means of the GRAPHIC EDITOR.

If the CNC shows the message **" Initialize? (ENTER / ESC) "**, it should be borne in mind that after pressing the ENTER key, all the information stored in memory and the machine parameters are initialized to default values indicated in the installation manual.

On the lower part of the screen the main CNC menu will be shown, it being possible to select the different operating modes by means of the softkeys F1 thru F7.

Whenever the CNC menu has more options than number of softkeys (7), the character "+" will appear in softkey f7. If this softkey is pressed the CNC will show the rest of the options available.

The options which the main CNC menu will show after turning it on, after pressing the key sequence SHIFT-RESET or after pressing the "MAIN MENU" softkey are:

EXECUTE Allows the execution of part programs in automatic or single block.

SIMULATE Allows simulation of parts programs in several modes.

EDIT Allows editing new and already-existing part programs.

JOG Allows manual control of the machine by means of the Control Panel keys.

TABLES Allows CNC tables relating to part programs (Zero Offsets, Tool Offsets, Tools, Tool Magazine and global or local arithmetic parameters) to be manipulated.

UTILITIES Allows program manipulation (copy, delete, rename, etc.)

STATUS It shows the CNC status and that of the DNC communication lines. It also lets activate and deactivate the communication with a PC through DNC.

DNC Allows communication with a computer via DNC to be activated or deactivated.

PLC Allows operation with the PLC (edit the program, monitor, change the status of its variables, access to the active messages, errors, pages, etc).

GRAPHIC EDITOR Allows, by means of a simple graphics editor, the creation of user-defined screens (pages), which can later be activated from the PLC, used in customized programs or presented when the unit is powered on (page 0).

MACHINE PARAMETERS Allows the machine parameters to be set to adapt the CNC to the machine.

DIAGNOSIS Makes a test of the CNC.

While the CNC is executing or simulating a part program it allows any other type of operating mode to be accessed without stopping the execution of the program.

In this way it is possible to edit a program while another is being executed or simulated.

It is not possible to edit the program which is being executed or simulated, nor execute or simulate two part programs at the same time.

2.1 HELP SYSTEMS

The CNC allows access to the help system (main menu, operating mode, editing of commands, etc.) at any time.

To do this, you must press the HELP key and the corresponding help page will be shown in the main window of the screen.

If the help consists of more than one page of information, the symbol indicating that this key can be pressed to access the following page or the indicating that it is possible to press this key to access the previous page.

The following help is available:

*** OPERATING HELP**

This is accessed from the operating mode menu, or when one of these has been selected but none of the options shown have been selected. In all these cases, the softkeys have a blue background color.

It offers information on the operating mode or corresponding option.

While this information is available on screen it is not possible to continue operating the CNC via the softkeys, it being necessary to press the HELP key again to recover the information which was on the main screen before requesting help and continuing with the operation of the CNC.

The help system can also be abandoned by pressing the ESC key or the MAIN MENU key.

*** EDITING HELP**

This is accessed once one of the editing options has been selected (part programs, PLC program, tables, machine parameters, etc.) In all these cases, the softkeys have a white background.

It offers information on the corresponding option.

While this information is available, it is possible to continue operating with the CNC.

If the HELP key is pressed again, the CNC analyzes if the present editing status corresponds to the same help page or not.

If another page corresponds to it, it displays this instead of the previous one and if the same one corresponds, it recovers the information which was in the main window before requesting help.

The help menu can also be abandoned after pressing the ESC key, to return to the previous operating option, or the MAIN MENU key to return to the main menu.

* CANNED CYCLES EDITING HELP

It is possible to access this help when editing a canned cycle.

It offers information on the corresponding canned cycle and an editing assistance for the selected canned cycle is obtained at this point.

For the user's own cycles a similar editing assistance can be obtained by means of a user program. This program must be prepared with screen customizing instructions.

Once all the fields or parameters of the canned cycle have been defined the CNC will show the information which exists in the main window before requesting help.

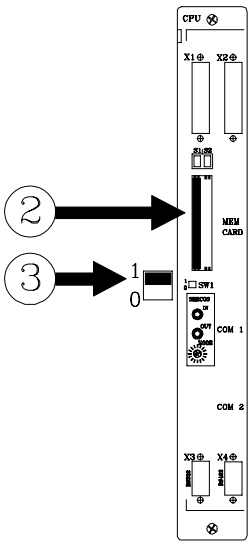
The canned cycle which is programmed by means of editing assistance will be shown in the editing window, and the operator can modify or complete this block before entering it in memory by pressing the ENTER key.

Editing assistance can be abandoned at any time by pressing the HELP key. The CNC will show the information which existed on the main window before requesting help and allows programming of the canned cycle to continue in the editing window.

The help menu can also be abandoned after pressing the ESC key, to return to the previous operating option, or the MAIN MENU key to return to the main menu.

2.2 SOFTWARE UPDATE

Procedure



1- Turn the CNC off

2.- Replace the memory card in "Slot A" with the one containing the new software version.

3.- Set the SW1 switch to "1".

4- Turn the CNC on.

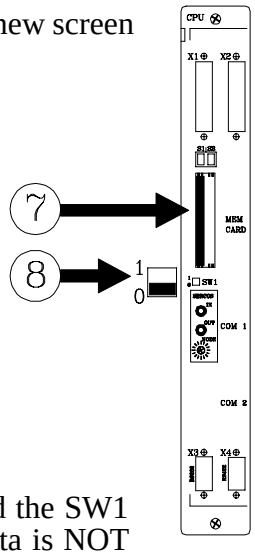
The screen will show the software updating page with the following information:

Installed version and New version
Checksum of the installed version and that of the new one.

5.- Press the [Update software] softkey

The CNC will display the various stages of the software updating process and their status.

When done with the updating process, the CNC will display a new screen with the steps to follow.



6.- Turn the CNC off

7.- Replace the memory card in "Slot A" with the "Memkey Card".

8.- Set the SW1 switch to “0”.

9- Turn the CNC on. The software version is now updated.

Notes: With the memory card that contains the software version, the CNC CANNOT executed anything.

If the CNC is turned on with the "Memkey Card" in and the SW1 switch set to "1", the CNC does not come on, but its data is NOT affected.

Warning:

Reinstall the CNC software when replacing the Hard Disc module
The CNC software and the Hard Disc module must be compatible.

3 *EXECUTE / SIMULATE*

The EXECUTE operating mode allows the execution of part programs in automatic mode or in single block mode.

The SIMULATE operating mode allows the simulation of part-programs in automatic or single block mode.

When selecting one of these operating modes, one must indicate the location of the part-program to be executed or simulated.

The part program may be stored in the CNC's internal RAM memory, in the "Memkey Card", in PC connected through serial line 1 or 2, or in the hard disk (HD module).

After pressing one of these softkeys, the CNC displayes the corresponding part-program directory.

The program may be selected by:

- Keying in its number and pressing [ENTER] or
- Positioning the cursor of the scren over the desired program and pressing [ENTER].

When wished to SIMULATE a part-program, the CNC will request the type of simulation to be carried out as shown on the next page.

The executing or simulating conditions (fist block, type of graphics, etc.) may be set before executing or simulating the part-program. These conditions may also be modified if the execution or simulation is interrupted.

To execute or simulate a part-program, press

Note: To switch to JOG mode once executed or simulated a part program (or a section of it), the CNC will maintain the machining conditions (type of movement, feedrates, etc.) selected while executing or simulating it.

The executing or simulating conditions (fist block, type of graphics, etc.) that may be set before executing or simulating the part-program are:

THEORETICAL PATH

Simulates the execution of the program without moving the axes, without taking tool radius compensation into consideration and without executing the auxiliary M, S, T functions.

G FUNCTIONS

Simulates the execution of the program without moving the axes, by executing the programmed G functions and without executing the auxiliary M, S, T functions.

G, M, S, T FUNCTIONS

Simulates the execution of the program without moving the axes, by executing the G functions and programmed auxiliary M, S, T functions.

MAIN PLANE

This option executes the selected part-program moving only the axes forming the main plane and executing the programmed M, S, T and G functions.

The axes movement will be carried out at top F0 feedrate regardless of the F0 value programmed. This feedrate can be modified by means of the feedrate override switch.

RAPID

Verifies the execution of the program by moving the axes, executing the G functions and the programmed auxiliary M, S, T functions.

Movements of the axes will be executed at the maximum feedrate permitted F0, regardless of the programmed F feedrates, thus allowing this feedrate to be varied by means of the FEEDRATE OVERRIDE switch.

Once the required program has been selected in the EXECUTION or SIMULATION modes and before pressing the key (cycle start) on the Operator Panel in order for the CNC to execute it, the following operations will be available:

BLOCK SELECTION

It allows selecting the block in which the execution or the simulation of the program will start.

STOP CONDITION

It allows selecting the block in which the execution or the simulation of the program will stop.

DISPLAY SELECTION

It allows the display mode to be selected.

MDI

It allows any type of block (ISO or high level) to be edited with programming assistance by means of softkeys.

Once a block has been edited and after pressing the key (cycle start), the CNC will execute this block without leaving this operating mode.

TOOL INSPECTION

Once the execution of the program has been interrupted, this option allows the tool to be inspected and changed should this be necessary.

GRAPHICS

This option carries out a graphic representation of the part during the execution or simulation of the selected part program.

It also allows selecting the type of graphic, the area to be displayed, the viewpoint and graphic parameters.

SINGLE BLOCK

Allows the part program to be executed one block at a time or continuously.

3.1 BLOCK SELECTION AND STOP CONDITION

The CNC will start to execute the required block from the first line of the program and will finish it when one of the program end functions M02 or M30 is executed.

If it is required to modify one of these conditions the **BLOCK SELECTION** and **STOP CONDITION** functions must be used.

BLOCK SELECTION

With this option it is possible to indicate the beginning block of the selected program execution or simulation. This cannot be used when the CNC is already executing or simulating the selected program.

When this option is selected, the CNC will show the selected program since the initial block must always belong to this program.

The operator must select with the cursor the block where the execution or simulation of the program will be started.

To do this, the cursor can be moved line by line with the up and down arrow keys or page by page with the page-up and page-down keys.

The “find” softkey options are also available:

BEGINNING: By pressing this key, the cursor will position at the first line of the program.

END: By pressing this key, the cursor will position at the last line of the program.

TEXT: With this function it is possible to search for a text or character sequence starting at the current cursor position.

When this softkey is pressed, the CNC requests the character sequence to be found.

Once this text has been keyed in, press the "**END OF TEXT**" softkey and the cursor will position over the first occurrence of the keyed text.

The found text will be highlighted and it will be possible to continue (by pressing "**ENTER**") with the search all along the program or quit by pressing either the "**ESC**" key or "**ABORT**" softkey.

The search can be done as many times as it is desired. Once searched to the end of the program, it will continue the search from the beginning.

When quitting the search mode, the cursor will be positioned at the last matching text found.

LINE NUMBER: After pressing this key, the CNC will request the number of the line to be found. Key in the desired line number and press **ENTER**. The cursor will, then, be positioned at the desired line.

Once the desired starting block is selected, press **ENTER** to validate it.

STOP CONDITION

With this option it is possible to indicate the final execution or simulation block of the selected program. This cannot be used when the CNC is already executing or simulating the selected program.

When selecting this option, the CNC will show the following softkey functions:

PROGRAM SELECTION

This option will be used when the final execution or simulation block belongs to another program or to a subroutine resident in another program.

The CNC shows the part-program directory of the RAM memory. Use the cursor to select the desired program and press ENTER.

Then, carry out the BLOCK SELECTION as described next.

BLOCK SELECTION

Use the cursor to select the last program block to be executed.

Use the up and down arrow keys or page by page with the page-up and page-down keys.

The "find" softkey options are also available:

BEGINNING: By pressing this key, the cursor will position at the first line of the program.

END: By pressing this key, the cursor will position at the last line of the program.

LINE NUMBER: After pressing this key, the CNC will request the number of the line to be found. Key in the desired line number and press **ENTER**. The cursor will, then, be positioned at the desired line.

Once the desired final block has been selected, press **ENTER** to validate it.

NUMBER OF TIMES

This function will be used to indicate that the execution or simulation of the selected program must stop after executing the "end block" a specific number of times.

When selecting this function, the CNC will request the number of times to be executed or simulated.

If a canned cycle or a call to a subroutine has been selected as the end block of the program, the CNC will stop after executing the complete canned cycle or the indicated subroutine.

If the selected block has a number of block repetitions, the program will stop after doing all the repetitions indicated.

3.2 DISPLAY SELECTION

With this option, it is possible to select the most appropriate display mode at any time even during execution or simulation of a part program.

The display modes available at the CNC and which can be selected with softkeys are:

STANDARD
POSITION
PART PROGRAM
SUBROUTINES
FOLLOWING ERRORS
USER
EXECUTION TIMES

All the display modes have a window at the bottom of the CRT which shows the history with the conditions in which machining is being done. The information shown is as follows:

F and **%** Programmed feedrate and selected feedrate OVERRIDE %.

S and **%** Programmed spindle speed and selected spindle OVERRIDE %

T Number of active tool.

D Number of active tool offset.

NT Number of the next tool

This field will be displayed when having a machining center and it will show the tool being selected but which is waiting for the execution of the M06 to make it active.

ND Tool offset number corresponding to the next tool.

This field will be displayed when having a machining center and it will show the tool being selected but which is waiting for the execution of the M06 to make it active.

S RPM Real speed of the spindle in RPM.

When working in M19 this indicates the position of the spindle in degrees.

G All displayable G functions which are active.

M All active M functions.

PARTC Parts counter. It indicates the number of consecutive parts executed with the same part-program.

Every time a new program is selected, this variable is reset to "0".

With this CNC variable (PARTC) it is possible to modify this counter from the PLC, from the CNC program and via DNC.

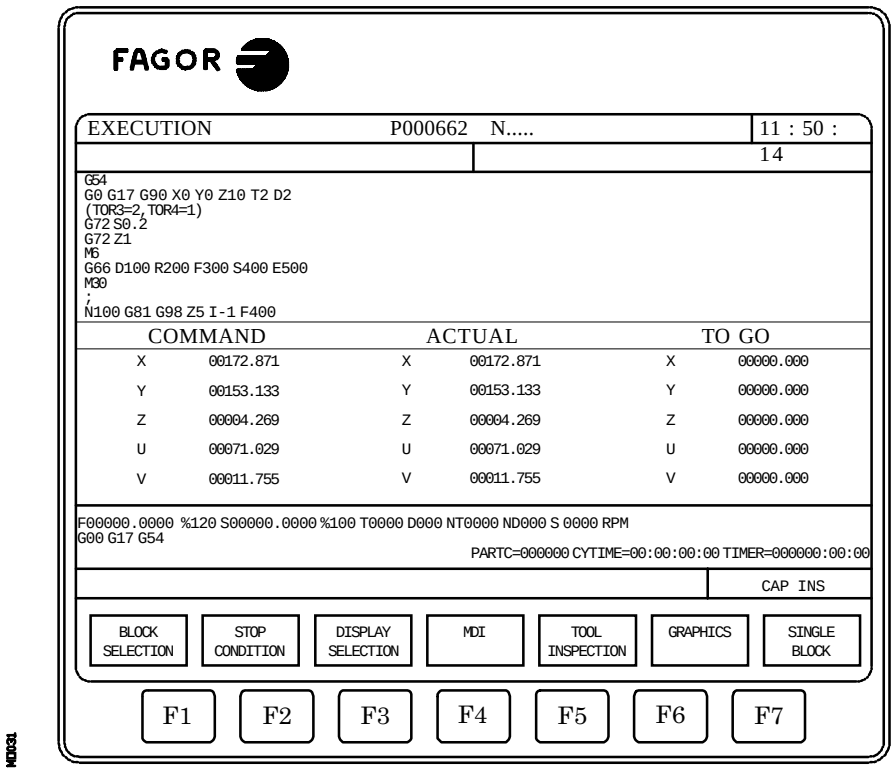
CYTIME Time elapsed during the execution of the part in “hours : minutes : seconds : hundredths of a second” format.

Every time a part-program execution starts, even when repetitive, this variable is reset to "0".

TIMER Time indicated by the PLC-enabled clock in “hours: minutes : seconds” format.

3.2.1 STANDARD DISPLAY MODE

This display mode is assumed by default on power-up and after the key sequence SHIFT-RESET and it shows the following fields or windows:



* A group of program blocks. The first of them is the block being executed.

* The axis coordinates, in real or theoretical values according to the setting of the "THEODPLY" machine parameter and the format defined with the axis machine parameter "DFORMAT".

Each axis is provided with the following fields:

COMMAND. Indicates the programmed coordinate or position value which the axis must reach.

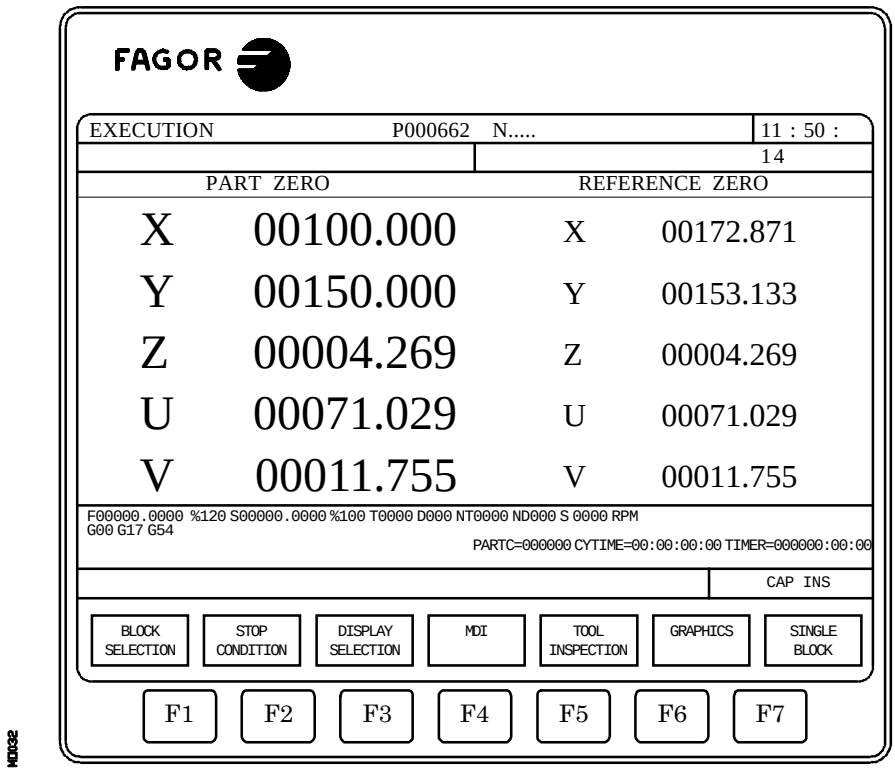
ACTUAL. Indicates the actual (current) position of the axis.

TO GO. Indicates the distance which is left to run to the programmed coordinate.

3.2.2 POSITION DISPLAY MODE

This display mode shows the position values of the axes.

This display mode shows the following fields or windows:



* The axis coordinates, in real or theoretical values according to the setting of the "THEODPLY" machine parameter and the format defined with the axis machine parameter "DFORMAT".

Each axis has the following fields:

PART ZERO This field shows the real axis position with respect to part zero.

MACHINE ZERO This field shows the real axis position with the respect to machine reference zero (home).

3.2.3. PART PROGRAM DISPLAY MODE

Displays a page of program blocks among which the block being executed is highlighted.

3.2.4. SUBROUTINE DISPLAY MODE

This display mode shows information regarding the following commands:

(RPT N10,N20) This function executes the program section between blocks N10 thru N20.

(CALL 25) This function executes subroutine number 25.

G87 ... This function the corresponding canned cycle.

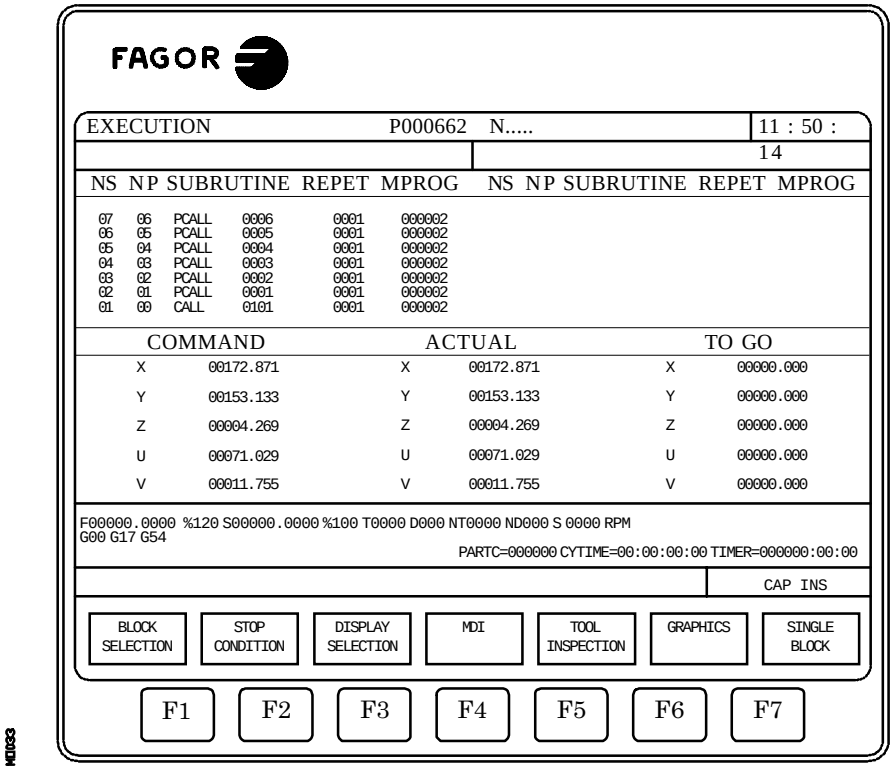
(PCALL 30) This function executes subroutine 30 in a local parameter level.

When this mode is selected, the following must be considered:

The CNC allows the definition and usage of subroutines which can be called upon from a main program or from another subroutine and this can, in turn, call upon a second one and so forth up to 15 nesting levels (each subroutine call represents a nesting level).

When the machining canned cycles: G66, G68, G69, G81, G82, G83, G84, G85, G86, G87, G88 and G89 are active, they use the sixth nesting level of local parameters.

This display mode shows the following fields or windows:



* Information on the subroutines which are active.

NS Indicates the nesting level (1-15) which the subroutine occupies.

NP Indicates the level of local parameters (1-6) in which the subroutine is executed.

SUBROUTINE Indicates the type of block which has caused a new nesting level.

Examples: (RPT N10,N20) (CALL 25) (PCALL 30) G87

REPT Indicates the number of times which remain to be executed.

For example, if (RPT N10, N20) N4 is programmed and is the first time that it is being executed, this parameter will show a value of 4.

M If an asterisk is shown (*) this indicates that a Modal subroutine is active in this nesting level, and this is executed after each movement.

PROG Indicates the program number where the subroutine is defined.

* The axis coordinates, in real or theoretical values according to the setting of the "THEODPLY" machine parameter and in the format determined by the axis machine parameter "DFORMAT".

  Each axis is provided with the following fields:

  COMMAND. Indicates the programmed coordinate or position which the axis must reach.

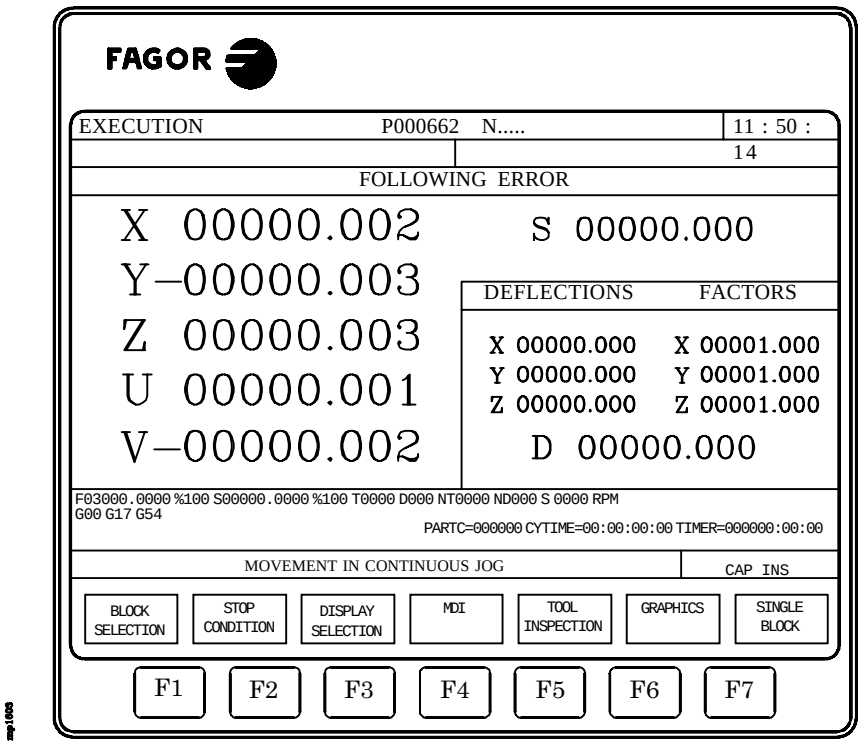
  ACTUAL. Indicates the actual (current) position of the axis.

  TO GO. Indicates the distance which is left to run to the programmed coordinate.

3.2.5 FOLLOWING ERROR DISPLAY MODE

This display mode shows the following error (difference between the theoretical value and the real value of their position) of the axes and the spindle.

Also, when having the tracing option, this mode shows, to the right of the screen, a window with the values corresponding to the tracing probe.



The display format is determined by the axis machine parameter “DFORMAT”.

The correction factors of the probe do not depend on the work units.

The display format for the probe deflections on each axis (X, Y, Z) as well as the total deflection "**D**" is set by axis machine parameter "**DFORMAT**".

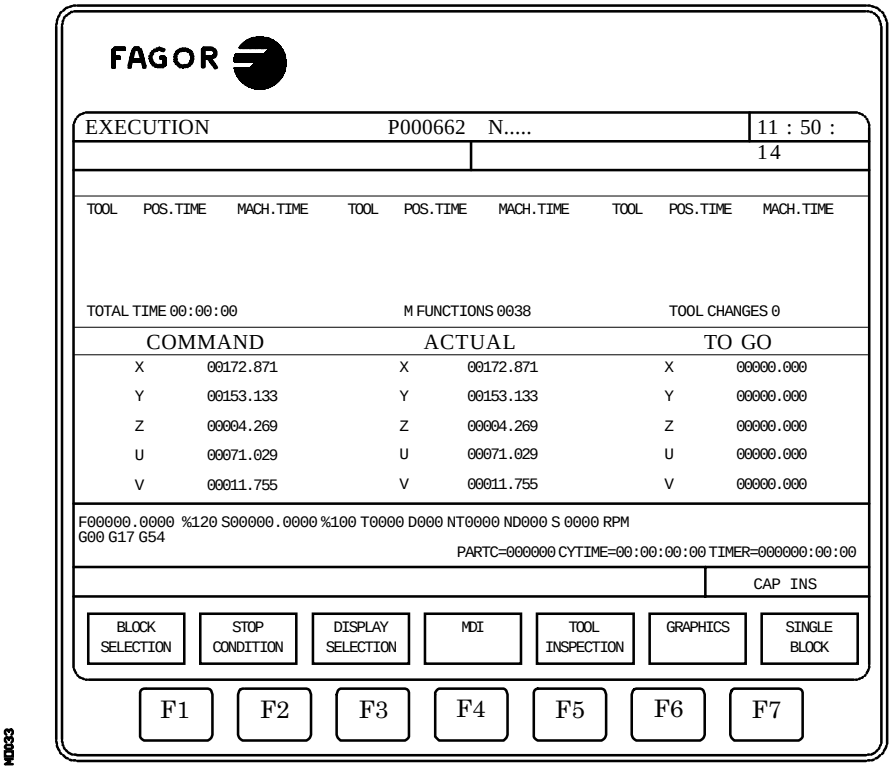
3.2.6 USER DISPLAY MODE

This option will execute the program which is selected by means of the general machine parameter “USERDPLY” in the user channel.

To quit this mode and return to the previous menu, press **ESC**.

3.2.7 EXECUTION TIME DISPLAY MODE

This option is available while simulating a part-program and it will display the following fields or windows:



* A display window shows the estimated program execution time at 100% of the programmed feedrate.

This display area shows the following information:

The time each tool (TOOL) takes to execute the positioning moves (POS.TIME) as well as the machining moves (MACH.TIME) indicated in the program.

The "TOTAL TIME" required to execute the complete program.

The "M FUNCTIONS" being executed in the program.

The number of "TOOL CHANGES" performed during the execution of the program.

* The position values for the axes of the machine.

  It must be borne in mind that the display format for the axes is established by machine parameter "**DFORMAT**" and that real or theoretical position values will be shown depending on the setting of machine parameter "**THEODPLY**".

  Each axis has the following fields:

  COMMAND. Indicates the programmed coordinate or position which the axis must reach.

  ACTUAL. Indicates the actual (current) position of the axis.

  TO GO. Indicates the distance which is left to run to the programmed coordinate.

3.3 MDI

This function is not available in the SIMULATION mode. Besides, if a program is being executed, it must be interrupted in order to access this function.

It is possible to execute any block (ISO or high level) and it provides information on the corresponding format via the softkeys.

Once the block has been edited and after the key has been pressed the CNC will execute this block without quitting this operating mode.

3.4 TOOL INSPECTION

This function is not available in the SIMULATION mode. Besides, if a program is being executed, it must be interrupted in order to access this function.

This operating mode allows all the machine movements to be controlled manually, and enabling the axis control keys on the Operator Panel (X+, X-, Y+, Y-, Z+, Z-, 4+, 4-, etc.).

Also, the CNC will show the softkeys to access the CNC tables, edit and execute a block in MDI as well as repositioning the axes of the machine to the position from where this function was called.

One of the ways to make the tool change is as follows:

* Move the tool to the required tool change position

  This move may be made by jogging the axes from the operator panel or in MDI.

* Gain access to CNC tables (tools. Tool offsets, etc.) in order to find another tool with the similar characteristics.

* Select, in MDI, the new tool as the active one.

* Make the tool change

  This operation will be performed depending on the type of tool changer used. It is possible to execute the tool change in MDI in this step.

* Return the axes to the position where the tool inspection began (REPOSITIONING).

* Continue executing the program ( )

Note: If during tool inspection, the spindle is stopped, the CNC will restart it in the same turning direction (M3 or M4) while repositioning.

The CNC offers the following options by means of softkeys:

MDI

Allows to edit blocks in ISO or high level (except those associated with subroutines) providing information on the corresponding format by means of softkeys.

Once the block has been edited and after the key has been pressed the CNC will execute this block without quitting this operating mode.

TABLES

Allows access to any of the CNC tables associated with part programs (Zero offsets, Tool offsets, Tools, Tool magazine, Global and Local Parameters).

Once the desired table has been selected, all editing commands will be available for its verification and modification.

In order to return to the previous menu the **ESC** key must be pressed.

REPOSITIONING.

Positions the axes at the point where tool inspection started.

Once this option is selected, the CNC will show the axes to be repositioned and will request the order in which they will move.

The "PLANE" softkey will appear for the main plane movements and another softkey for each one of the rest of the axes to be repositioned.

Once repositioning has been completed the key is pressed to continue with the execution of the rest of the program.

3.5 GRAPHICS

With this function it is possible to select the type of graphic to be used as well as to define all the parameters for the corresponding graphic display.

To do so, the CNC must NOT be executing or simulating a part program; otherwise, it must be interrupted.

Once the type of graphics has been selected and its parameters defined, this function can be accessed even during the execution or simulation of a part program should the type of graphic or any graphic parameters be changed

After selecting this function, the CNC will display the following softkey options:

* Type of graphic
* Display area
* Zoom
* Point of view
* Graphic parameters
* Clear Screen
* Deactivate graphics

One of the different ways that could be used to define graphics is the following:

1.- Define the DISPLAY AREA. It will depend on the dimensions of the part and its coordinate values will be referred to the part zero being currently active .

2.- Select the TYPE OF GRAPHICS to be displayed.

3.- Define the VIEWPOINT to be used. This option is available in types of graphics such as 3D and SOLID.

4.- Select the drawing colors to be used by means of the GRAPHIC PARAMETERS.

Once the part-program execution or simulation has been started, it is possible to interrupt it and define another type of graphic or select another graphic display area by means of the ZOOM option.

3.5.1 TYPE OF GRAPHICS

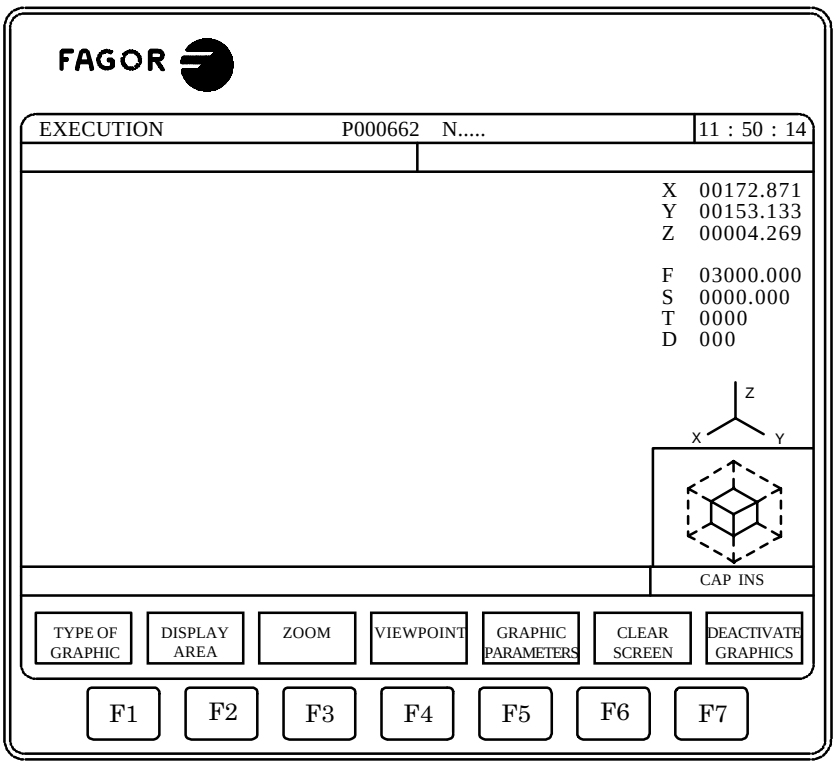
This CNC offers two types of graphics: line and solid graphics. They both are totally independent from each other in such a way that an execution or simulation performed in either one does not affect the other.

The CNC will show all the possible softkey options in order to select one of them.

The type of graphic will remain active until another type is selected or graphics are deactivated (with its corresponding softkey) or the CNC is turned off.

Every time a type of graphic is selected, the CNC recovers all the graphic conditions (zoom, graphic parameters and display area) which were active during the last type of graphic selected.

The selected type of graphics will display the following information to the right of the screen:



KD035

* The current real axes position. The tool position values will indicate the position of the tool tip.

* The axes feedrate (F) and the spindle speed (S) currently selected.

* The active tool (T) and tool offset (D).

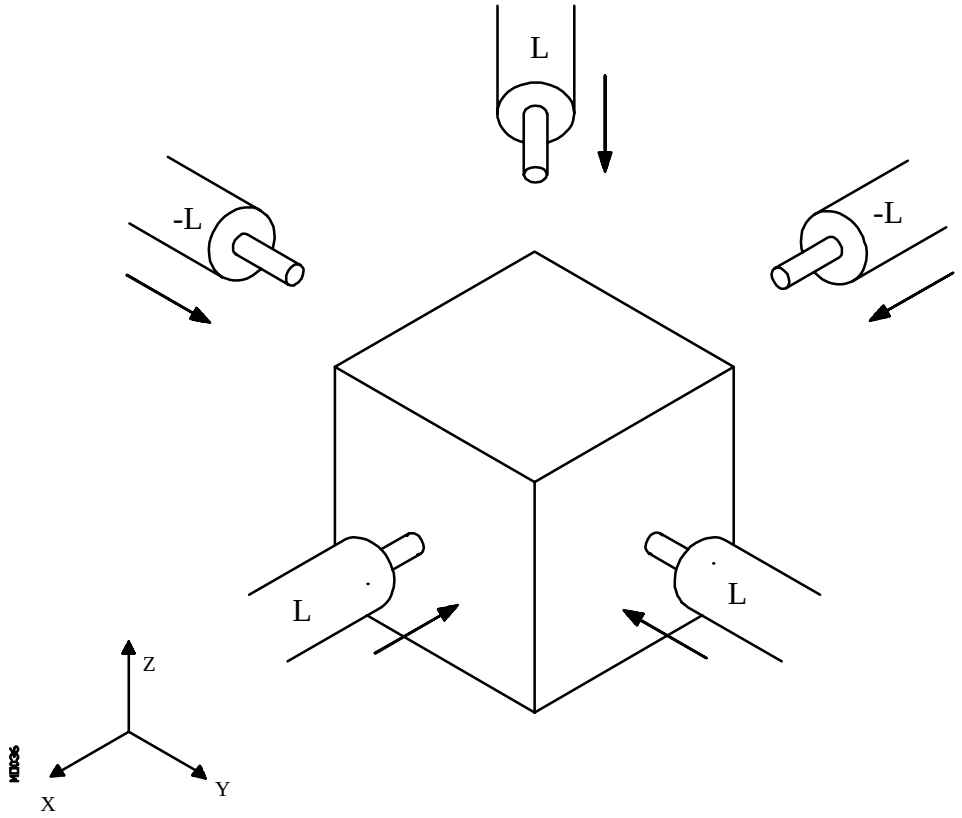
* The point of view used for the graphic display. It is defined by the X, Y, Z axes and it can be modified by means of the VIEWPOINT softkey.

* Two cubes or rectangles depending on the type of point of view selected.

  The cube, whose sides are colored, indicates the graphic area currently selected and the one drawn only with lines shows the size of display area being selected.

  When the point of view shows a single cube side or when the selected type of graphics corresponds to one of the XY, XZ or YZ planes, the CNC will display two rectangles indicating the graphic area (colored rectangle) and the display area being selected (non-colored rectangle).

This CNC will display all machining operations performed with the tool along either the X, Y or Z axis except when the tool is along the Z axis and the part is being machined on its negative side (in the -Z to +Z direction).



When simulating a part-program, the CNC analyzes the value assigned to the tool length in the corresponding tool offset.

If this value is positive, the graphic display is performed on the positive side of the part. (in the + to - direction) and if negative, it will be performed on the negative side of the part (in the - to + direction).

It must be borne in mind that the CNC will assume a value of L0 as positive. Also, if no tool has been defined during execution or simulation, the CNC will take L0 and R0 as default values.

LINE GRAPHICS

This type of graphics draws the tool path on the selected planes (XY, XZ, YZ) by means of color lines.

The possible types of line graphics are:

3D Displays a three-dimensional view of the tool path.

XY,XZ,YZ Display the tool path on the selected plane.

COMBINED VIEW This option divides the screen in four quadrants displaying in them the XY, XZ, YZ and 3D views simultaneously.

The generated graphics is lost in the following circumstances:

* When clearing the screen (softkey: CLEAR SCREEN).
* When deactivating graphics (softkey: DEACTIVATE GRAPHICS).
* When redefining a new display area (softkey: DISPLAY AREA).
* When selecting a new type of solid graphic (top view or solid)

SOLID GRAPHICS

This type of graphics offer the same information in two different ways: as a three-dimensional solid (SOLID) or as a section view of the part (SECTION VIEW).

When simulating or executing a program in any of these modes, it is possible to display its graphics in either mode.

The section view is usually drawn faster than the solid view, therefore, it is recommended to first run the program in section view and then switch to solid graphics. The end result will be the same.

The graphic generated after executing or simulating a program is lost in the following instances:

* When clearing the screen (softkey CLEAR SCREEN).
* When deactivating the graphics (softkey DEACTIVATE GRAPHICS).
* When redefining a new display area (softkey DISPLAY AREA).
* When selecting a new line graphics (3D, XY, XZ, YZ, Combined).

SECTION VIEW

This option displays a section view of the part on the XY plane drawn in different shades of gray which indicate the depth of the part.

The other plane views are also shown (XZ and YZ) which correspond to the sections indicated by the vertical and horizontal indicator lines.

These vertical and horizontal indicator lines can be moved left and right or up and down respectively by means of the corresponding arrow keys.

These indicator lines can be moved at any time even while executing or simulating the part-program and the CNC will display "live" the new sections corresponding to the new indicator line positions.

Once the execution or simulation has finished or it has been interrupted, the CNC redraws the section view in order to achieve a better color definition and better sense of depth.

This type of graphics will not show the machining operations performed with the tool positioned along the X or Y axis but only when positioned along the Z axis. However, when switching to SOLID, afterwards, all machining operations will be shown.

SOLID

This option shows a three-dimensional block which will be "machined" as the part-program is being run.

If no tool has been selected while executing or simulating the part-program, the CNC will assume a default tool offset value of L0, R0. With these values, the CNC will only show the programmed tool path and the block will not be "machined" since the tool is assumed to have no radius (R0).

The screen refresh is done periodically depending on the simulation speed and always from left to right regardless of the movement direction of the tool.

It must be borne in mind that when executing or simulating a new program (other than the current one), it will be "machined" over the existing "already-machined" block. However, a new "unmachined" block can be obtained by deleting the screen with the CLEAR SCREEN softkey.

3.5.2 DISPLAY AREA

In order to use this option, the CNC must not be executing or simulating a part-program. If so, it must be interrupted.

With this option it is possible to define the display area by assigning the desired values to maximum and minimum coordinates for each axis. These coordinate values must be referred to part zero.

This maximum and minimum coordinate assignment will be done in the windows displayed to the right of the screen which show their current values.

Use the up and down arrow keys to select the desired field whose value is to be changed.

Once the desired values for all the desired fields have been keyed in, press **ENTER** to validate them.

To quit this mode without making any changes, press **ESC**.

While SOLID GRAPHICS or SECTION VIEW is selected, it must be borne in mind that if a new display area is defined, the CNC will reset the graphic representation returning to its initial status, “unmachined”.

In linear graphics (3D, XY, XZ, YZ, combined) there is a softkey [optimum area] which redefines the display area that contains, in all planes, all the tool paths already executed.

3.5.3 ZOOM

In order to use this option, the CNC must not be executing or simulating a part-program. If so, it must be interrupted.

With this option, it is possible to enlarge or reduce the graphics display area. It cannot be used in either COMBINED VIEW or SECTION VIEW types of graphics.

When selecting this option, the CNC will show a window superimposed on the current graphics and another one over the drawing at the lower right-hand side of the screen. These new windows indicate the new display area being selected.

Use the [zoom +] and [zoom-] keys to either enlarge or reduce the size of the new display area and the arrow keys to move the zoom window around to the desired location on the screen.

By pressing the softkey [INITIAL VALUE], it assumes the values set by means of [DISPLAY AREA]. The CNC shows that value, but it does not quit the zoom mode.

Once the new display area has been defined, press **ENTER** to validate the new values.

Press **ESC** to quit this ZOOM mode without making any changes to the initial values.

Every time a Zoom is carried out in 3D, XY, XZ, YZ graphics, it redraws the machining executed up to that point. If the number of points to be redrawn exceeds the amount of memory reserved for it, only the last points will be redrawn and the older ones will be lost.

When zooming into a solid graphics, the drawing will be initialized showing a new unmachined 3D block.

3.5.4 VIEWPOINT

In order to use this option, the CNC must not be executing or simulating a part-program. If so, it must be interrupted.

This option can be used with any three-dimensional graphics (3D, COMBINED VIEW or SOLID) and it allows to change the point of view (perspective) of the part by shifting the X, Y and Z axes.

When selecting this option, the CNC will highlight the current viewpoint on the right-hand side of the screen.

Use the right and left arrow keys to rotate the XY plane around the Z axis up to 360°.

Use the up and down arrow keys to tilt the Z axis up to 90°.

Once the new viewpoint has been selected, press **ENTER** to validate it.

If SOLID GRAPHICS was selected before or it is selected again, the CNC will refresh the screen showing the same part but from the new viewpoint (with new perspective).

When the selected type of graphics is 3D or COMBINED VIEW, the CNC will maintained the current drawing. The new viewpoint will be applied when executing the next blocks. These blocks will be drawn over the existing graphics. However, the screen can be cleared by using the CLEAR SCREEN softkey in order to start drawing with an “unmachined” part.

To quit this mode without making any changes, press **ESC**.

3.5.5 GRAPHIC PARAMETERS

This function can be used any time, even during part program execution or simulation:

With this function it is possible to modify the simulation speed and the colors used to draw the tool paths.

The modifications made to any parameter are immediately assumed by the CNC and can be made during the execution or simulation of the part program.

The softkey options displayed by the CNC are:

SIMULATION SPEED

With this option it is possible to modify the percentage of the speed used by the CNC to execute the part programs in the simulation modes.

The CNC will display a window at the top right-hand side of the screen indicating the current % of simulation speed.

This value can be modified by using the right and left arrow keys. Once the desired value is selected, press **ENTER** to validate the new value.

Press **ESC** to quit this function without making any changes to this field.

It is also possible to change the simulation speed while it is redrawing after a zoom. This lets you check the machining of a particular operation.

PATH COLORS

With this option it is possible to modify the colors used to draw the various tool paths in the execution and simulation modes. They can only be used in line graphics XZ. The available parameters are:

The color for representing rapid moves
The color for representing path without compensation
The color for representing path with compensation
The color for representing threading
The color for representing canned cycles

The CNC will show a series of windows for the definition of graphics parameters.

Among the various colors to choose from, there is a black or “transparent” one. If this one is chosen for a particular path, this path will not be displayed on the screen.

If any of them is to be modified, first select the corresponding window using the up and down keys and then use right and left arrow keys to select the desired color.

Once the desired colors have been selected, press **ENTER** to validate the new choices or **ESC** to ignore the changes and leave this function with the original values intact.

SOLID COLORS

With this option it is possible to modify the colors used in the three-dimensional solid graphics. These colors will be considered when in execution or simulation and will only be used in SOLID graphics mode. The available parameters are the following:

Color for the external X side
Color for the external Y side
Color for the external Z side
Color for the internal X side, machined side
Color for the internal Y side, machined side
Color for the internal Z side, machined side

The CNC will show to the right of the screen a series of windows to select these parameters indicating as well the colors currently selected.

Among the various color choices, the black one indicates that the machining operations done with this color will not be shown graphically (invisible).

To modify any of these parameters, select the corresponding field by using the up and down arrow keys and use the right and left arrow keys to select the color within the desired field or window.

Once the desired colors for the desired solid sides have been selected, press **ENTER** to validate them.

Press **ESC** to quit this color selection mode without making any changes to the original settings.

3.5.6 CLEAR SCREEN

In order to use this function, no part program may be in execution or simulation. If this is the case, it must be interrupted.

Erases the screen or graphic representation shown.

If the solid graphic mode is selected, it will return to its initial status showing the unmachined part.

3.5.7 DEACTIVATE GRAPHICS

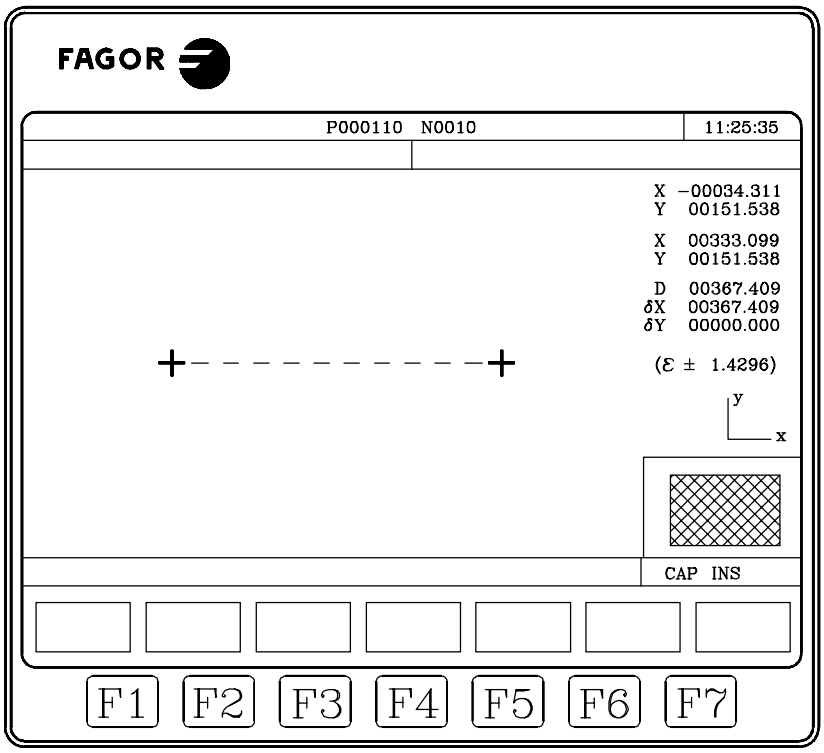
It allows the graphic representation to be deactivated at any time, even during execution or simulation of a part program.

To activate this function again, the “GRAPHICS” softkey must be pressed again. To do this, the CNC must not be executing or simulating a part program. If this is the case, it must be interrupted.

3.5.8 MEASURE

To use this function, a "Line Graphics" (planes XY, XZ or YZ) must be selected and the CNC must not be executing or simulating the part-program. If it is, it must be interrupted.

Once this function is selected, the CNC shows the following information on the screen:



The center of the CRT shows a dotted line with two cursors, the section to be measured. Also, the right-hand side of the screen shows:

* The coordinates of those two cursors with respect to part-zero.

* The distance "D" between them and the components of this distance along the axes of the selected plane " δ X" and " δ Y".

* The cursor step " ε " corresponding to the selected display area. It is given in the work units, millimeters or inches.

The CNC shows the selected cursor and its coordinates in red.

To select the other cursor, press the "+" or "-" key. The CNC shows the new selected cursor and its coordinates in red.

To move the selected cursor, use the up, down, right and left arrow keys.

Also, with the keystroke sequences: Shift-Up arrow, Shift-Down arrow, Shift-Right arrow and Shift-Left arrow, it is possible to move the cursor to the corresponding end.

To quit this command and return to the graphics menu, press [**ESC**]

Also, if [I] is pressed, the CNC exits this work mode and returns to the graphics menu.

3.6 SINGLE BLOCK

When actuating on this option, the CNC toggles between single block mode and continuous run mode. This function can be used at any time, even during the execution or simulation of a part program.

If the single block mode is selected, the CNC will only execute one line of the program every time the is pressed.

The upper window of the screen will show the selected mode of operation. If continuous execution, no message will appear and if SINGLE BLOCK, it will display the message: SINGLE BLOCK.

4. *EDIT*

This operating mode will be used to edit, modify or look at a part-program stored in the CNC's RAM memory.

To edit a part-program stored in the "Memkey Card" (CARD A) or in the hard disk (HD), it must be previously copied into RAM memory.

To edit a part-program, enter the program number (up to 6 digits) from the keyboard or by selecting it with the cursor from the CNC's part-program directory and then pressing **ENTER**.

Move the cursor on the screen line by line with the “up and down” arrow keys or page by page with the “page up” and “page down” keys.

Once the program number has been entered, the CNC will display the softkeys for the following options:

EDIT *(See section 4.1)*
To edit new lines in the selected program.

MODIFY *(See section 4.2)*
To modify an existing line of the program.

FIND *(See section 4.3)*
To search a string of characters within a program.

REPLACE *(See section 4.4)*
To replace a string of characters with another.

DELETE BLOCK *(See section 4.5)*
To delete a block or group of blocks.

MOVE BLOCK *(See section 4.6)*
To move a block or group of blocks within a program.

COPY BLOCK *(See section 4.7)*
To copy a block or group of blocks to another program position.

COPY TO PROGRAM *(See section 4.8)*
To copy a block or group of blocks into a different program.

INCLUDE PROGRAM *(See section 4.9)*
To insert the contents of another program into the one currently selected.

EDITOR PARAMETERS *(See section 4.10)*
To select the editing parameters (automatic numbering and axes for Teach-in editing).

4.1 EDIT

With this option it is possible to edit new lines or blocks of the selected program.

Select with the cursor the block after which the new ones will be added and press the softkey corresponding to one of the available editing modes.

CNC LANGUAGE ..... *(See section 4.1.1)*
The program is edited in ISO code or high level language.

TEACH-IN ..... *(See section 4.1.2)*
The machine is jogged to the desired position and, then, the new axis position may be assigned to the block.

INTERACTIVE ..... *(See section 4.1.3)*
Editing mode assisted by the CNC.

PROFILES ..... *(See section 4.1.4)*
To edit a new profile
After defining the known profile data, the CNC generates its corresponding ISO-coded program.

PROFILE SELECTION
To modify an existing profile.
The CNC requests the first and last blocks of the profile.
Once they are both defined, the CNC will show the corresponding graphics.
Section 4.1.4 describes how to operate with the profile.

USER
When selecting this option, the CNC will execute, in the user channel, the customizing program selected by general machine parameter "USEREDIT".*(See section 4.1.1)*
This is edited in ISO-code or high level language.

4.1.1 EDITING IN CNC LANGUAGE

A program will be edited block by block and each block can be written either in ISO code or high level language or it can be just a program comment.

Once this option has been selected, the softkeys will change colors and they will appear over white background showing the information corresponding to the type of editing possible at that point.

Also, editing help will be available at any time by just pressing the **HELP** key. To quit this help mode, press **HELP** again.

If ESC is pressed while editing a block, the block editing mode is abandoned and the block currently being edited will not be added to the program.

Once the block has been edited, press **ENTER.** This new block will be added to the program after the one indicated by the cursor.

The cursor will position over the new edited block and the editing area (window) will be cleared so another block can be written.

To quit the block editing mode, press **ESC** or **MAIN MENU.**

4.1.2 TEACH-IN EDITING

It is basically identical to the previous option (editing in CNC language), except what regards the programming of position coordinate values.

This option shows the current position values of each one of the axes of the machine.

It permits to enter the axes position values from the CNC keyboard (as when editing in CNC language) or, also, use the TEACH-IN editing format as described next.

* Jog the machine axes with the jogging keys or with the electronic handwheel up to the desired position.

* Press the softkey corresponding to the axis to be defined.

* The CNC will assign to this axis its current physical position as the program position value.

Either position value programming methods can be used at any time while defining a block.

When the block being edited has no information (empty editing area or window), the **ENTER** key may be pressed in which case the CNC will generate a new block with the current position values of the axes.

This block will be added automatically to the program and it will be inserted after the block indicated by the cursor.

The cursor will position over the new edited block and the editing area will be cleared so another can be written.

When the position values of <u>all</u> the axes are <u>not</u> to be programmed in this fashion, the CNC permits to select the desired axes. To do this, in this operating mode and within the "EDITOR PARAMETERS" option there is a soft key for "**TEACH-IN AXES**"

4.1.3 INTERACTIVE EDITOR

This editor leads the operator through the program editing process by means of questions he/she will answer.

This type of editing offers the following advantages:

* No knowledge of the CNC programming language is required.

* The CNC only admits the data it is requesting, thus no erroneous data can be entered.

* The programmer has, at all times, the appropriate programming aide by means of screens and messages.

When selecting this option, the CNC displays in the main window, a series of graphic options selectable by softkey.

If the selected option has more menus, the CNC will keep showing new graphic options until the desired one is selected.

From this moment, the information corresponding to this option will appear in the main window and it will start requesting the data necessary to program it.

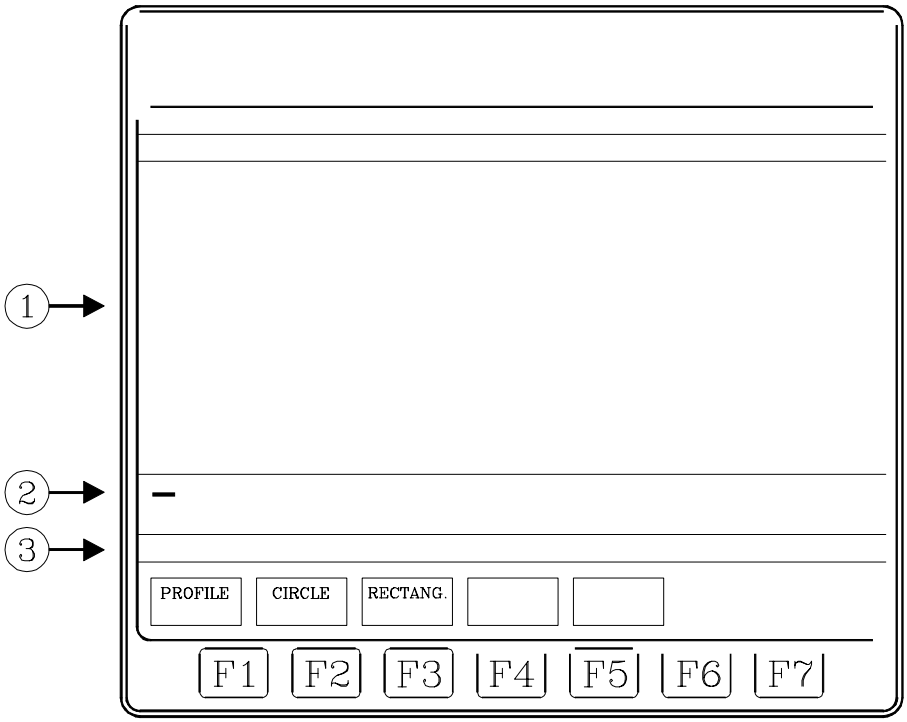
As the requested data is entered, the editing window will show, in CNC language, the block being edited.

The CNC will generate all necessary blocks and it will add them to the program once the editing of this option is done and it will insert them after the one indicated by the cursor.

The main window will show again the graphic options corresponding to the main menu being possible to continue editing the program.

4.1.4 PROFILE EDITOR

When selecting this option, the CNC displays the following fields or windows:



1.- Window showing the graphic representation of the profile being edited.

2.- Editing window showing the new generated block in CNC language.

3.- Area for editing messages.

4.- Display area

Indicates the area of the plane shown in the graphic representation of the profile. Indicated by the maximum and minimum position values of each axis.

The way to select this display are is described later on.

5.- Display area for the profile section currently selected for editing or modifying.

It may be the starting block, straight line, a clockwise arc or a counterclockwise arc.

6.- Display area for additional information. It shows a series of parameters for internal use and whose meanings are:

Et : Total elements of the profile
Ec : Complete elements
Ni : Number of data entered
Nr : Number of required data

4.1.4.1 OPERATION WITH THE PROFILE EDITOR

Several profiles may be edited without quitting the profile editor. To edit a profile, proceed as follows:

1.- Select a point of the profile as its beginning point.

2.- Break the profile into straight and curve sections.

If the profile has corner roundings, chamfers, tangential entries or exits, take one of the following actions:

- Treat them as individual sections when having enough information to define them.

- Ignore them when defining the profile and, once done defining the whole profile, select the corners showing those characteristics and enter the corresponding radius value.

CONFIGURATION

Use the [abscissa axis] and [ordinate axis] softkeys to select the editing plane.

The Autozoom function indicates whether the CNC recalculates the graphics display area or not when the edited lines go beyond it.

PROFILE

For editing any profile.

CIRCLE

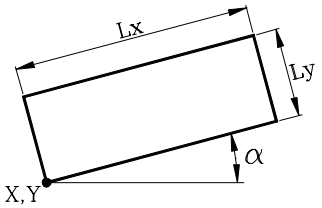
For a quick circular profile definition. If the starting point (X,Y) is not defined, the CNC assumes one.

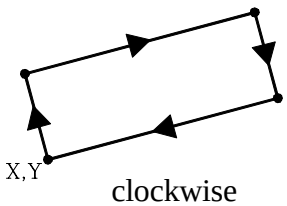
The [Profile Direction] softkey indicates whether the profile is programmed clockwise or counterclockwise. This data is very important for later modifications and profile intersection. Every time this softkey is pressed, the text at the top of the middle right window changes.

STRAIGHTANGLE

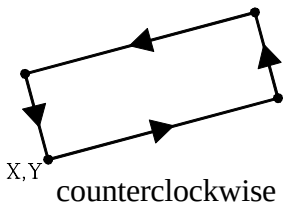
For a quick straight angular profile definition.

The [Profile Direction] softkey indicates whether the profile is programmed clockwise or counterclockwise. This data is very important for later modifications and profile intersection. Every time this softkey is pressed, the text at the top of the middle right window changes.





clockwise



counterclockwise

A straight angular profile is defined with a single command, but the CNC internally breaks into 4 straight segments.

4.1.4.2 PROFILE EDITING

When pressing the [PROFILE] softkey, the CNC requests the starting point of the profile. To define it, use the corresponding softkeys.

For example, if when working in the XY plane the new desired starting point is (20,50):

[X] 20 [ENTER]
[Z] 50 [ENTER]

The values may be set by means of a numeric constant or by means of any expression. Examples:

X 100
X 10 * cos 45
X 20 + 30 * sine 30
X 2 * (20 + 30 * sine 30)

Once the starting point has been set, press the [VALIDATE]

The CNC will show a filled circle in the graphics area to indicate the starting point of the profile.

Also, the softkeys will show the following options:

[STRAIGHT LINE]	To edit a straight section.
[CLOCKWISE ARC]	To edit a clockwise arc.
[COUNTERCLOCKWISE ARC]	To edit a counterclockwise arc.
[CORNERS]	To insert roundings, chamfers, tangential entries and exits.
[MODIFY]	To modify the starting point. Modify any profile element, even the type of element (straight line, clockwise or counterclockwise arc) Insert a new element (straight line or arc) in any position of the profile. Delete any profile element. Add a new additional text to any section of the profile. Modify the display area.
[NEW PROFILE]	To edit a new profile.
[FINISH]	It must be pressed when all the sections of the profile have been defined. It must be indicated whether the edited profile or profiles must be saved or not. The CNC quits the profile editor and adds to the program the ISO code corresponding to the profile just edited .

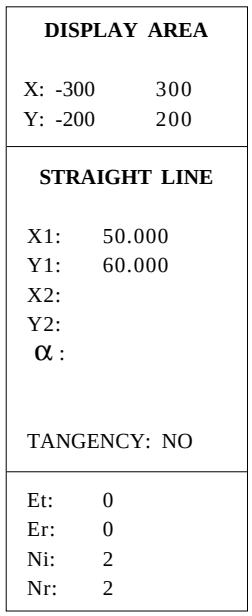
4.1.4.3 DEFINITION OF A STRAIGHT SECTION

When pressing the [STRAIGHT LINE] softkey, the CNC displays the data shown on the right margin of this page.

X1, Y1	Coordinates of starting point of the line. They cannot be modified because they correspond to the last point of the previous section.
X2, Y2	Coordinates of the end point of the section.
α	Angle of the line referred to the abscissa axis.
TANGENCY	Indicates whether the line to be drawn is tangent to the previous section or not.

All these parameters need not be defined, but all the known ones should be defined.

To define a parameter, press the corresponding softkey, key in the desired value and press [ENTER].



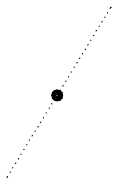
The value may be defined by a numeric constant or by any expression.

Examples:
X 100
X 10 * cos 45
X 20 + 30 * sine 30
X 2 * (20 + 30 * sine 30)

Once all known parameters are set, press the [VALIDATE] softkey and the CNC will show the defined section, if possible.

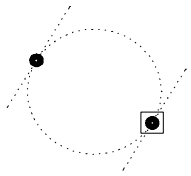
If there is not enough data to show the section, the CNC will show a dotted line indicating its orientation.

Example
X1=0
Y1=0
X2
Y2
α = 60



If there are more than one possibility, all the possible options will be shown and the desired one (framed in red) must be selected using the right and left arrow keys.

Example
X1
Y1
X2
Y2
α = 60
TANGENCY = YES



Use the up and down arrow keys to choose whether all the possible options are shown or only the one framed in red.

Once the desired option is selected, press [ENTER] for the CNC to assume it.

4.1.4.4 DEFINITION OF A CIRCULAR SECTION

When pressing the [CLOCKWISE ARC] or [COUNTERCLOCKWISE ARC] softkey, the CNC displays the data shown on the right margin of this page.

X1, Y1	Coordinates of the starting point of the arc. They cannot be modified because they correspond to the last point of the previous section.
X2, Y2	Coordinates of the end point of the arc.
XC, YC	Coordinates of the arc center.
XC, YC	Radius of the arc.
TANGENCY	Indicates whether the arc to be drawn is tangent to the previous section or not.

All these parameters need not be defined, but all the known ones should be defined.

DISPLAY AREA

X: -300 300
Y: -200 200

CLOCKWISE ARC

X1: 50.000
Y1: 60.000
X2:
Y2:
XC:
YC:
RA
TANGENCY: NO

Et: 0
Er: 0
Ni: 2
Nr: 2

To define a parameter, press the corresponding softkey, key in the desired value and press [ENTER].

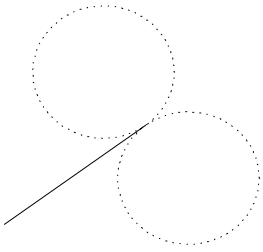
The value may be defined by a numeric constant or by any expression.

Examples:
X 100
X 10 * cos 45
X 20 + 30 * sine 30
X 2 * (20 + 30 * sine 30)

Once all known parameters are set, press the [VALIDATE] softkey and the CNC will show the defined section, if possible.

If there are more than one possibility, all the possible options will be shown and the desired one (framed in red) must be selected using the right and left arrow keys.

Example
X1 = 40
Y1 = 30
X2
Y2
XC
YC
RA = 20
TANGENCY = YES



Use the up and down arrow keys to choose whether all the possible options are shown or only the one framed in red.

If there is not enough data to show the section, the CNC waits for more data in order to solve the profile.

Once the desired option is selected, press [ENTER] for the CNC to assume it.

4.1.4.5 CORNERS

When selecting this option, the CNC shows the following option softkeys:

Rounding	For rounding any corners of the profile.
Chamfer	For adding chamfers at any corner of the profile.
Tangential Entry	To add a tangential tool entry when machining.
Tangential Exit	To add a tangential tool exit at the end of the machining operation.

When selecting one of these, one of the corners of the profiles will appear highlighted.
To select another corner of the same profile, use the up/down and left/right arrow keys.
To select a corner of another profile, use the [page up] and [page down] keys.

To define the rounding, enter the rounding radius and press [ENTER].

To define the chamfer, enter the chamfer radius and press [ENTER].

To define the tangential entry, enter the radius of the path that the tool has to follow when doing a tangential entry and press [ENTER].

To define the tangential exit, enter the radius of the path that the tool has to follow when doing a tangential exit and press [ENTER].

To quit the CORNER mode, press [ESC].

4.1.4.6 MODIFY

When selecting this option, the CNC shows the following softkey options:

Starting Point	To modify the starting point of the profile.
Modify element	To modify any element of the profile, even the type of element (straight lines, clockwise or counterclockwise arcs).
Insert element	To insert a new element (straight line or arc) in any position of the profile.
Delete element	To delete any element of the profile.
Additional Text	To add additional text to any section of the profile.
Configuration	To add a new editing plane or redefine the Autozoom option.
Display area	To change the display area.

When selecting one of these options, one of the profile elements will be highlighted.
To select another element of the same profile, use the up/down and left/right arrow keys.
To select an element of another profile, use the [page up] and [page down] keys.

Starting point

- Select the desired element. The CNC shows the values used to define it.
- Select the starting point of the desired profile. The CNC shows the values used to define it.
- Modify the desired values and press the [VALIDATE] softkey.

Modify element

- Select the desired element. The CNC shows the values used to define it.
- It is possible to: change the type of section (straight or arc), redefine the existing data, define a new data or delete an existing one.
- To delete data, press the softkey that defines it and press [ESC]
- Once the element has been modified, press the [VALIDATE] softkey. The CNC recalculates the new profile with the data used to define that section and the next one (tangency, angle, etc.).

Insert element

- Select the point, or corner, after which the new one is to be inserted.
- Select the type of section (straight or arc), define it and press the [VALIDATE] softkey.
- The CNC recalculates the new profile with the data used to define that section and the next one (tangency, angle, etc).

Delete element

- Select the element to be deleted and confirm the command.
- The CNC recalculates the new profile.

Additional text

- Select the desired element. The CNC shows the ISO code corresponding to that section in the editing area.
- Add the desired text. Functions F, S, T, D, M or program comments may be added.
- Press the [VALIDATE] softkey.

Display area

When selecting this option, the following softkey options are shown:

- [Zoom +] to enlarge the image on the screen.
- [Zoom -] to reduce the image on the screen.
- [Optimum area] to show the full profile on the screen.
- The display area may be moved around with the [left arrow], [right arrow], [up arrow] and [down arrow] keys.
- Press the [VALIDATE] softkey. The CNC updates the values indicated in the upper right-hand window (DISPLAYED AREA).

To quit the MODIFY mode, press [ESC].

4.1.4.7 FINISH

This softkey must be pressed once all the sections of the profile have been defined.

The CNC will try to calculate the requested profile by previously solving all the unknowns.

If it finds several possibilities for certain sections, the CNC will show them for each section and the desired option (framed in red) will have to be chosen using the right and left arrow keys.

Once the whole profile has been solved, the CNC will show the code of the part program currently being edited.

The ISO-coded program for the edited profile is contained between these lines:

```
;************************** START **********************
;************************** END **********************
```

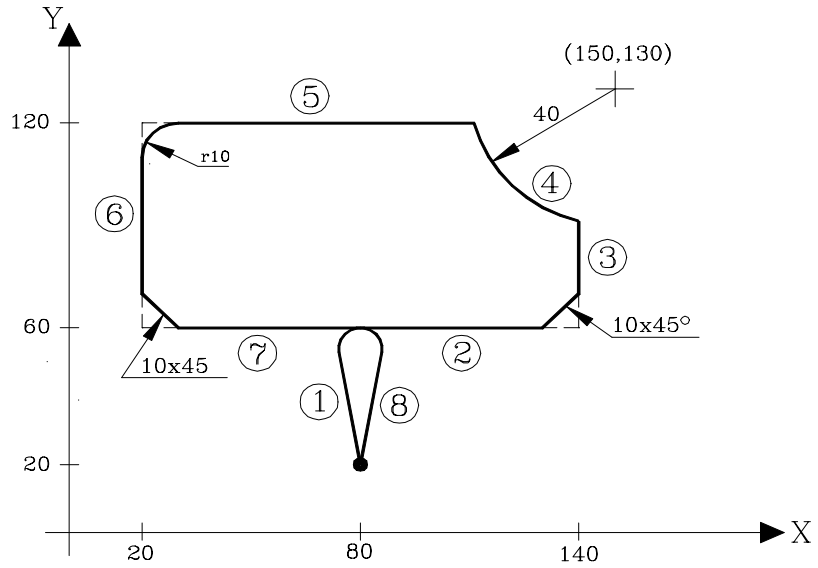
If a profile cannot be solved due to lack of data, the CNC will issue the corresponding error message.

Warning:

When pressing the [FINISH] softkey, the CNC quits the profile editor and adds to the program the ISO-code corresponding to the profile just edited.

To quit the profile editor without changing the part-program, press [ESC] and the CNC will request confirmation of this command.

4.1.4.8 EXAMPLES OF PROFILE DEFINITION



Profile definition without rounding, chamfers, tangential entries or exits.

Abscissa and ordinate of the starting point		X = 80 Y = 20
Section 1	STRAIGHT LINE	X = 80 Y = 60
Section 2	STRAIGHT LINE	X = 140 Y = 60
Section 3	STRAIGHT LINE	α = 90
Section 4	CLOCKWISE ARC	XC = 150 YC = 130 Radius = 40
		The CNC shows the possible intersections between sections 3 and 4. Select the correct one.
Section 5	STRAIGHT LINE	X = 20 Y = 120 α = 180
		The CNC shows the possible intersections between sections 4 and 5. Select the correct one.
Section 6	STRAIGHT LINE	X = 20 Y = 60
Section 7	STRAIGHT LINE	X = 80 Y = 60
Section 8	STRAIGHT LINE	X = 80 Y = 20

Adapt the image to the screen

Select the DISPLAY AREA option and press the [OPTIMUM AREA] softkey.

Definition of roundings, chamfers and tangential entries and exits.

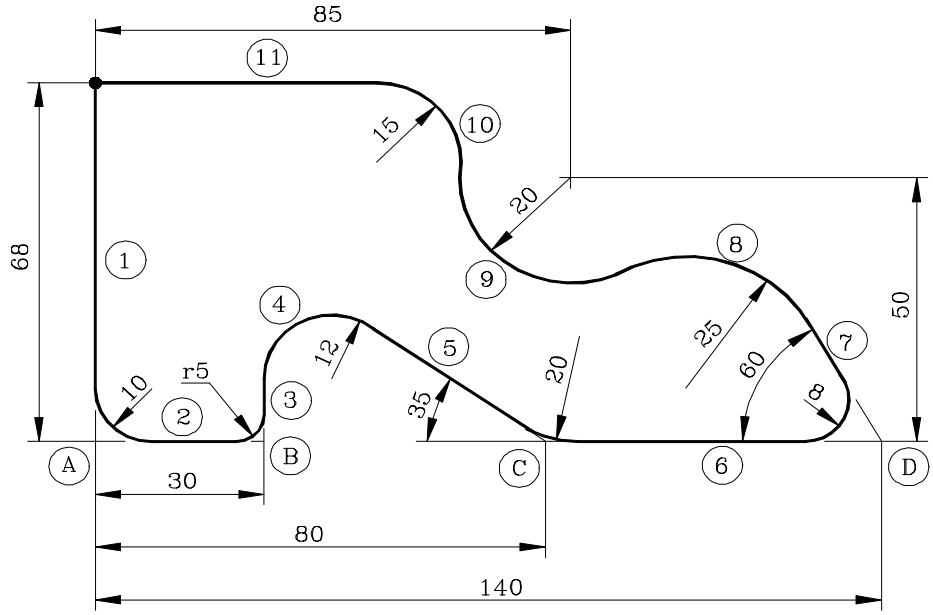
Select the MODIFY option and define:

CHAMFER Select corner 2-3 and press ENTER. With Radius = 10
ROUNDING Select corner 5-6 and press ENTER. With Radius = 10
CHAMFER Select corner 6-7 and press ENTER With Radius = 10
TANGENTIAL ENTRY
Select corner 1-2 and press ENTER. With Radius = 5
TANGENTIAL EXIT
Select corner 7-8 and press ENTER. With Radius = 5

Press ESC to quit the Modify option.

End of the editing process

Press the [FINISH] softkey. The CNC quits the profile editing mode and the shows the ISO-coded program that has been generated.



Profile definition without rounding

Abscissa and ordinate of the starting point			X = 0	Y = 68	
Section 1	STRAIGHT LINE		X = 0	Y = 0	
Section 2	STRAIGHT LINE		X = 30	Y = 0	
Section 3	STRAIGHT LINE		α = 90		
Section 4	CLOCKWISE ARC		RA=12	Tangent = Yes	
Section 5	STRAIGHT LINE	X = 80	Y = 0	α = -35	Tangent = Yes
The CNC shows the possible solutions for section 4. Select the correct one.					
Section 6	STRAIGHT LINE			X = 140	Y = 0
Section 7	STRAIGHT LINE			α = 120	
Section 8	COUNTERCLOCKWISE ARC			RA=25	Tangent = Yes
Section 9	CLOCKWISE ARC	XC = 85	YC = 50	RA=20	Tangent = Yes
The CNC shows the possible solutions for section 8. Select the correct one.					
Section 10	COUNTERCLOCKWISE ARC			RA=15	Tangent = Yes
Section 11	STRAIGHT LINE	X = 0	Y = 68	α = 180	Tangent = Yes

The CNC shows the possible solutions for section 10. Select the correct one.

Adapt the image to the screen

Select the DISPLAY AREA option and press the [OPTIMUM AREA] softkey.

Rounding definition

Select the MODIFY option and define:

ROUNDING	Select the A corner and press ENTER	With Radius = 10
ROUNDING	Select the B corner and press ENTER	With Radius = 5
ROUNDING	Select the C corner and press ENTER	With Radius = 20
ROUNDING	Select the D corner and press ENTER	With Radius = 8

Press ESC to quit the Modify option.

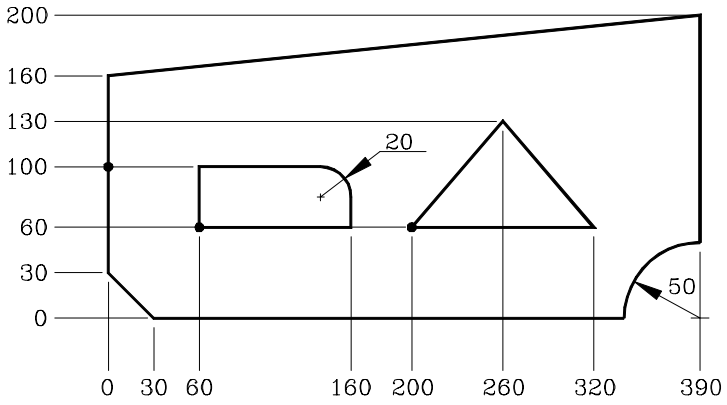
End of the editing process

Select the FINISH softkey. The CNC quits the profile editing mode and shows the ISO coded program that has been generated.

Example of how to define a profile and modify it later:

Configuration

Abscissa axis: X Ordinate axis: Y Autozoom: Yes Validate



Profile (outside profile)

Starting point X 0 Y 100 Validate
Straight X 0 Y 0 Validate
Straight X 340 Y 0 Validate
Clockwise arc Xf 390 Yf 50 R 50 Validate
(choose the right arc)
Straight X 390 Y 200 Validate
Straight X 0 Y 160 Validate
Straight X 0 Y 100 Validate

New Profile (rectangle)

Rectangle X 60 Y 60 Lx 100 Ly 40 Validate

New Profile (triangle)

Profile
Starting point X 200 Y 60 Validate
Straight X 320 Y 60 Validate
Straight X 260 Y 130 Validate
Straight X 200 Y 60 Validate

Corners (roundings and chamfers)

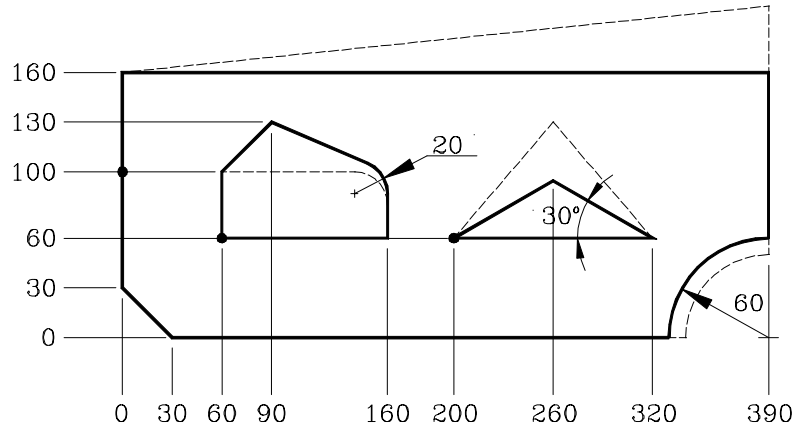
Chamfer

Select the first profile with the page up/down keys.
Select the lower left corner with the up&down and left/right arrow keys Enter
Chamfer 30
Escape

Rounding

Select the second profile (rectangle) the upper right corner Enter
Radius 20
Escape
Escape

Modify



(modify first profile)

Modify element
Select the lower line on the first profile Enter
Straight X 330 Validate (choose arc)
Modify - Modify element
Select the arc of the lower right corner Enter
Clockwise arc Yf 60 R 60 Validate (choose arc)
Modify - Modify element
Select right line Enter
Straight Y 160 Validate

(modify second profile)

Modify - Insert element
Select the second profile
Select the theoretical upper right corner Enter
Straight X 90 Y 130 Validate

(modify third profile)

Modify - Modify element.
Select the right side of the triangle on the third profile Enter
Straight Y Escape (to delete) Angle 150 Validate

Finish.

4.2 MODIFY

This option permits modifying the contents of a selected program block.

Before pressing this softkey, select with the cursor the block to be modified .

Once this option is selected, the softkeys will change their color showing their type of modifying option over a white background.

Also, it is possible to get more editing assistance by pressing **HELP**. Press **HELP** again to exit the editing assistance mode.

By pressing **ESC**, the information corresponding to that block and which was shown in the editing area will be cleared. It will then be possible to modify its contents again.

To quit the block modifying mode, press **CL** or **ESC** to clear the editing window and then press **ESC** again. This way, the selected block will not be modified.

Once the block contents have been modified, press **ENTER** so the new contents replace the old ones.

4.3 FIND

This option is used to find a specific text within the selected program.

When selecting this option, the softkeys will show the following options:

BEGINNING This softkey positions the cursor over the first program block which is then selected quitting the "find" option.

END This softkey positions the cursor over the last program block which is then selected quitting the "find" option.

TEXT With this function it is possible to search a text or character sequence starting from the block indicated by the cursor.

When this key is selected, the CNC requests the character sequence to be found.

When the text is defined, press the "END OF TEXT" softkey and the cursor will be positioned over the first occurrence of that text.

The search will begin at the current block.

The text found will be highlighted being possible to continue with the search or to quit it.

Press **ENTER** to continue the search up to the end of the program. It is possible to search as many times as wished and when the end of the program is reached, it will start from the first block.

Press the "EXIT" softkey or the **ESC** key to quit the search mode. The cursor will be positioned where the indicated text was found last.

LINE NUMBER After pressing this key, the CNC requests the number of the block to be found. After keying in the desired number and pressing **ENTER**, the cursor will position over that block which will then be selected quitting the search mode.

4.4 REPLACE

With this function it is possible to replace a character sequence with another throughout the selected program.

When selecting this option, the CNC requests the character sequence to be replaced.

Once the text to be replaced is indicated, press the "WITH" softkey and the CNC will request the character sequence which will replace the previous one.

Once this text is keyed in, press the "END OF TEXT" softkey and the cursor will be positioned over the first occurrence of the searched text.

The search will begin at the current block.

The found text will be highlighted and the following softkey options will appear:

REPLACE Will replace the highlighted text and will continue the search from this point to the end of the program.

If no more occurrences of the text to be replaced are found, the CNC will quit this mode.

If another occurrence of the text is found, it will be highlighted showing the same "replacing" or "not replacing" options.

DO NOT REPLACE Will not replace the highlighted text and will continue the search from this point to the end of the program.

If no more occurrences of the text to be replaced are found, the CNC will quit this mode.

If another occurrence of the text is found, it will be highlighted showing the same "replacing" or "not replacing" options.

TO THE END This function will automatically replace all the matching text from the current block to the end of the program without offering the option of not replacing it.

ABORT This function will not replace the highlighted text and it will quit the "find and replace" mode.

4.5 DELETE BLOCK

With this function it is possible to delete a block or group of blocks.

To delete only one block, just position the cursor over it and press **ENTER**.

To delete a group of blocks, indicate the first and last blocks to be deleted. To do so, follow these steps:

* Position the cursor over the first block to be deleted and press the “INITIAL BLOCK” softkey.

* Position the cursor over the last block to be deleted and press the “FINAL BLOCK” softkey.

  If the last block to be deleted is also the last one of the program, it can also be selected by pressing the “TO THE END” softkey.

* Once the first and last blocks are selected, the CNC will highlight the selected blocks requesting confirmation to delete them.

4.6 MOVE BLOCK

With this option it is possible to move a block or group of blocks by previously indicating the first and last blocks to be moved. To do so, follow these steps:

* Position the cursor over the first block to be moved and press the “INITIAL BLOCK” softkey.

* Position the cursor over the last block to be moved and press the “FINAL BLOCK” softkey.

  If the last block to be moved is also the last one of the program, it can also be selected by pressing the “TO THE END” softkey.

  To move only one block, the “initial block” and the “final block” will be the same one.

* Once the first and last blocks are selected, the CNC will highlight the selected blocks requesting confirmation to move them.

  Then, indicate the block after which this group of blocks must be placed.

* Press the “START OPERATION” softkey to carry out the move.

4.7 COPY BLOCK

With this option it is possible to copy a block or group of blocks by previously indicating the first and last blocks to be copied. To do so, follow these steps:

* Position the cursor over the first block to be copied and press the “INITIAL BLOCK” softkey.

* Position the cursor over the last block to be copied and press the “FINAL BLOCK” softkey.

  If the last block to be copied is also the last one of the program, it can also be selected by pressing the “TO THE END” softkey.

  To copy only one block, the “initial block” and the “final block” will be the same one.

* Once the first and last blocks are selected, the CNC will highlight the selected blocks requesting confirmation to copy them.

  Then, indicate the block after which this group of blocks must be placed.

* Press the “START OPERATION” softkey to carry out this command.

4.8 COPY TO PROGRAM

With this option it is possible to copy a block or group of blocks of one program into another program.

When selecting this option, the CNC will request the number of the destination program where the selected block or blocks are to be copied. After entering the program number press **ENTER**.

Next, indicate the first and last blocks to copy by following these steps:

* Position the cursor over the first block to be copied and press the "INITIAL BLOCK" softkey.

* Position the cursor over the last block to be copied and press the "FINAL BLOCK" softkey.

  If the last block to be copied is also the last one of the program, it can also be selected by pressing the "TO THE END" softkey.

  To copy only one block, the "initial block" and the "final block" will be the same one.

* Once the first and last blocks are selected, the CNC will highlight the selected blocks and will execute the command.

  If the destination program already exists, the following options will be displayed:

* Write over the existing program. All the blocks of the destination program will be erased and will be replaced by the copied blocks.

* Append (add) the copied blocks behind the ones existing at the destination program.

* Abort or cancel the command without copying the blocks.

4.9 INCLUDE PROGRAM

With this option it is possible to include or merge the contents of another program into the one currently selected.

Once this option is selected, the CNC will request the number of the source program to be merged. After keying in that number press **ENTER**.

Next, indicate with the cursor the block after which the source program will be included.

Finally, press the “START OPERATION” softkey to execute the command.

4.10 EDITOR PARAMETERS

With this option it is possible to select the editing parameters used in this operating mode.

The options or parameters available are described here and they are selected by softkeys.

4.10.1 AUTONUMBERING

With this option it is possible to have the CNC automatically number (label) the blocks after the one being edited.

Once this option is selected, the CNC will display the "ON" and "OFF" softkeys to either activate or deactivate this function.

Once this function is activated, the following options will appear on the CRT:

STEP After pressing this softkey, Enter the desired numbering step between two consecutive blocks and press **ENTER**.

The default value is 10.

STARTING After pressing this softkey, Enter the starting block number to be used on the next block to be edited.

The default value is 0.

When setting both parameters, select the STEP first and then the STARTING block number.

Example: STEP = 12, STARTING= 56;
generated blocks: N56, N68, N80,...

Warning:

This function will not number the already existing blocks.

4.10.2 AXES SELECTION FOR TEACH-IN EDITING

Remember that in the TEACH-IN editing mode, the following feature is available:

When the block being edited has no information (editing area empty), the "**ENTER**" key can be pressed. In this case, the CNC will generate a new block with the current position values of the axes.

The option described here, permits the selection of the axes whose position values will be automatically entered in said block.

After pressing the "**TEACH-IN AXES**" softkey, the CNC shows all the axes of the machine.

The operator must eliminate, pressing the corresponding softkeys, the axis or axes not desired. Every time a softkey is pressed, the CNC will eliminate the corresponding axis displaying only the selected ones.

To end this operation, press "**ENTER**".

The CNC will assume from now on and whenever editing in TEACH-IN, the selected axes. To change those values, access this option again and select the new axes.

5. *JOG*

This mode of operation will be used whenever the manual control of the machine is desired.

Once this mode of operation is selected, the CNC allows the movement of all the axes by means of the axes control keys (X+, X-, Y+, Y-, Z+, Z-, 4+, 4-) located on the operator panel, or by means of the electronic handwheel (if available).

This mode of operation offers the following softkey options:

With the **MDI** option it is possible to modify the machining conditions (type of moves, feedrates, etc.) being selected. Also, the CNC will maintain the ones selected in this mode when switching to **"EXECUTION"** or **"SIMULATION"** modes.

This operating mode offers the following softkey options:

REFERENCE SEARCH

With this option it is possible to perform a home search on the desired axis or axes.

The CNC offers two ways to search the machine reference (home):

* Using the subroutine associated with function G74. The number of this subroutine will defined by the general machine parameter “REFSUB”.

* By selecting the axis or axes to be referenced.

Once the Reference search function is selected, the CNC will show a softkey for each axis and the softkey “ALL”.

If the “ALL” softkey is selected, the CNC will highlight (in reverse video) the names of all axes and after pressing the [↵] key, it will execute the subroutine associated with G74.

On the other hand, to search the reference anywhere from one to all axes at once (without executing the associated subroutine), the softkeys corresponding to those axes must be pressed.

After pressing each softkey, the CNC will highlight the name of the selected axis.

If an unwanted axis has been selected, press **ESC** to cancel that selection and return to select “REFERENCE SEARCH”.

Once all the desired axes have been selected, press [↵].

The CNC will start the home search by moving all selected axes at once until the home reference switches for all axes are pressed and, from then on, the CNC will continue the home search one axis at a time.

Warning:

When searching home using the "ALL" softkey, the CNC will maintain the part zero or zero offset active at the time. However, if the axes have been selected one by one, the CNC will assume the "home" position as the new part zero.

PRESET

With this function it is possible to preset the desired axis position value.

Once this option is selected, the CNC will show the softkey corresponding to each axis.

After pressing the softkey of the corresponding axis to be preset, the CNC will request the position value to be preset with.

Press **ENTER** after the value has been keyed in so the new value is assumed by the CNC.

TOOL CALIBRATION

With this function it is possible to calibrate the length of the selected tool by using a part of known dimensions for this purpose.

Before pressing this softkey, the tool to be calibrated must be selected.

The tool calibration will be performed on the selected axis by means of the G15 function as longitudinal axis (by default: the Z axis).

When using a probe for tool calibration, the following machine parameters must be properly set: **"PRBXMIN", "PRBXMAX", "PRBYMIN", PRBYMAX", "PRBZMIN" , "PRBZMAX**" and **"PRBMOVE"**.

Tool calibration without a probe

Follow these steps:

* Press the softkey corresponding to the axis to be calibrated.

* The CNC will request the position value of the known part at the touch point.

  Once this value has been keyed in, press **ENTER** for this value to be assumed by the CNC.

* Jog the tool with the jog-keys (X+, X- Y+, Y-, Z+, Z-, 4+, 4-) until touching the part.

* Press the "**LOAD**" softkey corresponding to this axis.

  The CNC will perform the necessary calculations and it will assign the new value to the selected tool length offset.

Tool calibration with a probe

It may be done in two ways, as described in "calibration without a probe" or as follows:

* Press the softkey which indicates the direction of the tool calibration along the longitudinal axis.

* The CNC will move the tool at the feedrate indicated by the machine parameter for that axis "**PRBFEED**" until touching the probe.

  The maximum distance the tool can move is set by machine parameter "**PRBMOVE**".

* When the tool touches the probe, the CNC stops the axis and, after making the pertinent calculations, it will assign the new tool length value to its corresponding offset.

MDI

With this function it is possible to edit and execute a block (ISO or high-level) providing the necessary information by means of softkeys.

Once the block has been edited, press [I] to execute it without leaving this operation mode.

Warning:

When searching home "G74", the CNC will maintain the part zero or zero offset active at the time.

USER

When selecting this option, the CNC will execute, in the user channel, the program whose number is indicated in the general machine parameter “USERMAN”.

To quit its execution and return to the previous menu, press **ESC**.

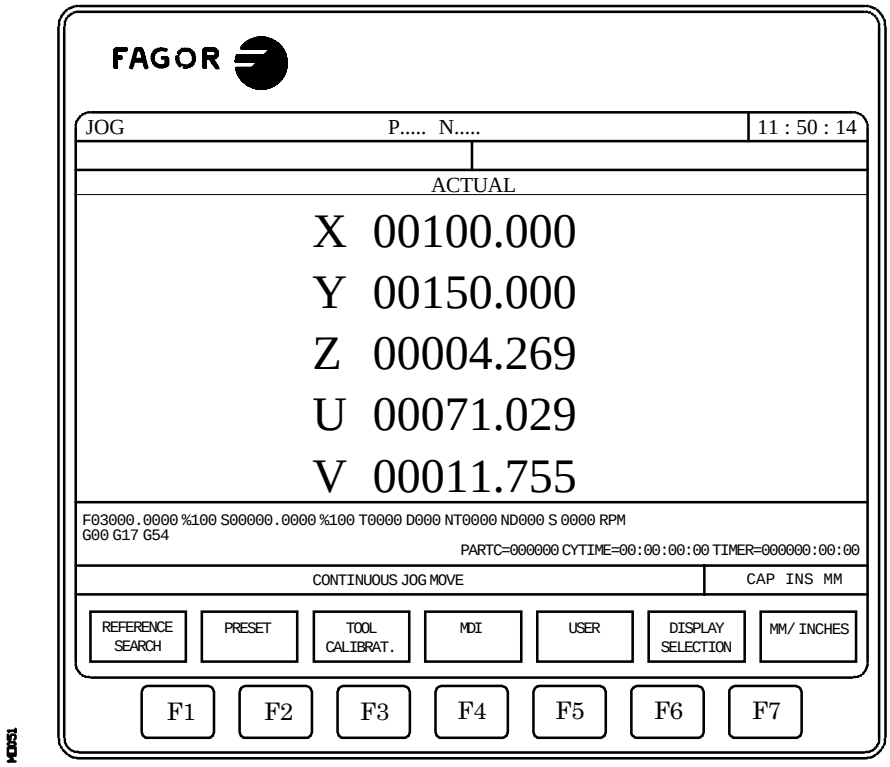
DISPLAY SELECTION

With this function it is possible to monitor the PLC by pressing the corresponding softkey. Once in that mode, operate as described in the chapter regarding the monitoring of the PLC.

It is also possible to select with the corresponding softkey one of the following position value (coordinate) displays:

ACTUAL

When selecting this option, the CNC will show the current position of the axes with respect to part zero.

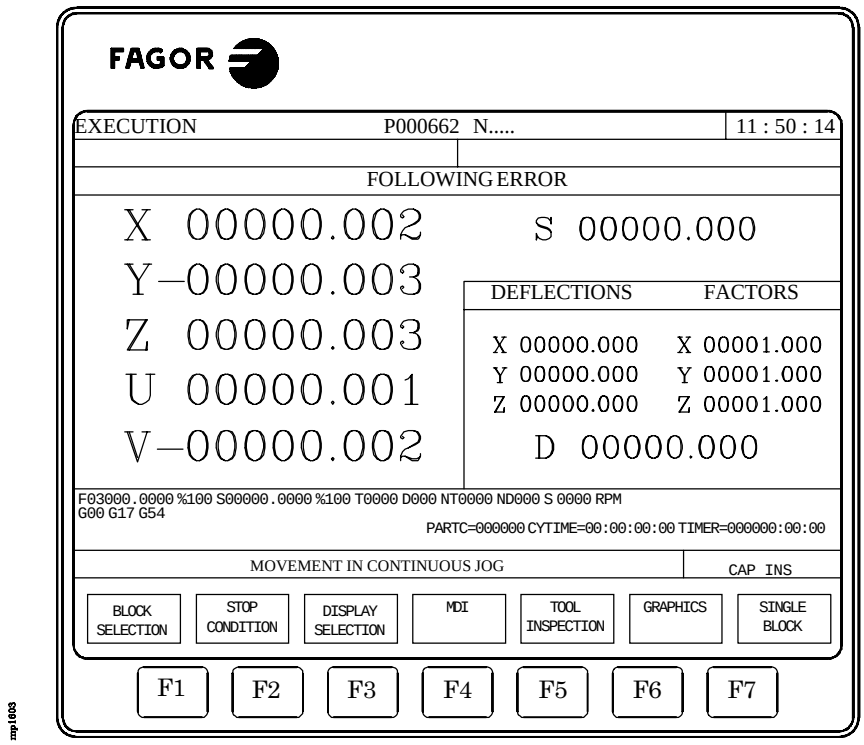


MDR51

FOLLOWING ERROR

When selecting this option, the CNC will show the following error (difference between the theoretical and real positions of the axes) for each axis and the spindle.

Also, when having the tracing option, this mode shows, to the right of the screen, a window with the values corresponding to the tracing probe.



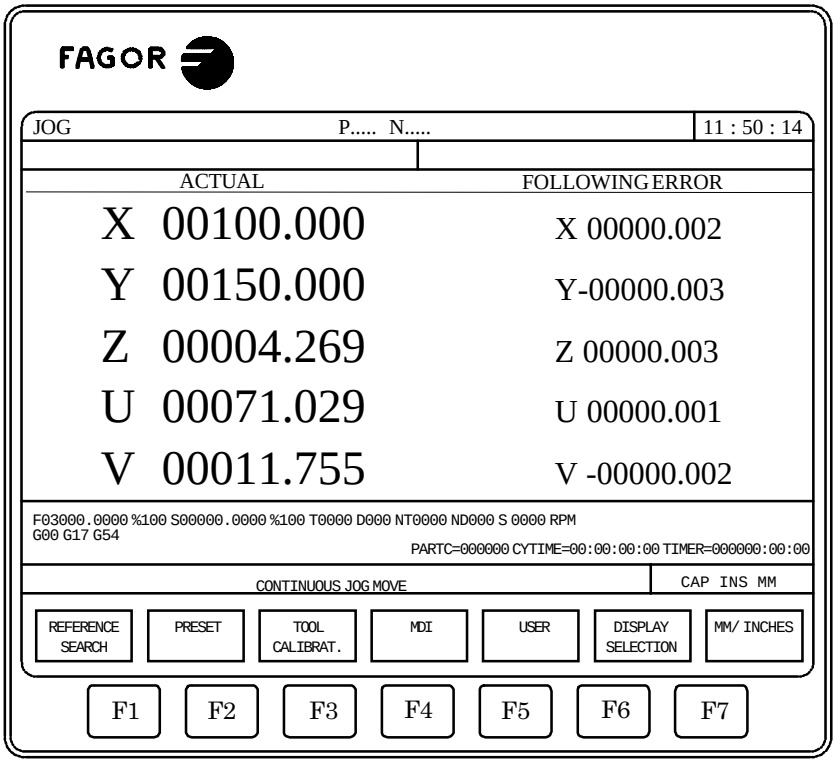
The display format is determined by the axis machine parameter “DFORMAT”.

The correction factors of the probe do not depend on the work units.

The display format for the probe deflections on each axis (X, Y, Z) as well as the total deflection "**D**" is set by axis machine parameter "**DFORMAT**".

ACTUAL AND FOLLOWING ERROR

When selecting this option, the CNC will show both the actual axes positions and their following errors.



MM/INCHES

This softkey toggles the display units for the linear axes from millimeters to inches and vice versa.

The lower right-hand window will indicate which units are selected at all times.

Note that this switching obviously does not affect the rotary axes which are shown in degrees.

5.1 JOGGING THE AXES

5.1.1 CONTINUOUS JOG

Once the % override of the jogging feedrate (indicated by axis-machine parameter "JOGFEED") has been selected with the switch at the Operator Panel , press the jog keys corresponding to the desired axis and to the desired jogging direction (X+, X-, Y+, Y-, Z+, Z-, 4+, 4- etc.).

The axes can be jogged one at a time and in different ways depending on the status of the general logic input "LATCHMAN":

* If the PLC sets this mark low, the axes will be jogged while pressing the corresponding Jog key.

** If the PLC sets this mark high, the axes will be jogged from the time the corresponding Jog key is pressed until the key is pressed or another jog key is pressed. In this case, the movement will be transferred to the axis corresponding to the new jog key.

If while jogging an axis, the key is pressed, the axis will move at the feedrate established by machine parameter "G00FEED" for this axis as long as this key stays pressed. When releasing this key, the axis will recover the previous feedrate (with its override %).

5.1.2 INCREMENTAL JOG

It allows to jog the selected axis in the selected direction an incremental step selected by the Feedrate Override switch and at the feedrate indicated by machine Parameter for that axis "JOGFEED".

The available positions are: 1, 10, 100, 1000 and 10000 corresponding to display resolution units.

Example:

Display format: 5.3 in mm or 4.4 in inches

Switch position	Movement
1	0.001 mm or 0.0001 inch
10	0.010 mm or 0.0010 inch
100	0.100 mm or 0.0100 inch
1000	1.000 mm or 0.1000 inch
10000	10.000 mm or 1.0000 inch

The maximum permitted step is 10 mm or 1 inch regardless of the selected display format (for example: 5.2 in mm or 4.3 in inches).

After selecting the desired incremental move at the switch, if a jog key is pressed (X+, X-, Y+, Y-, Z+, Z-, 4+, 4- etc.), the corresponding axis will move the selected distance in the selected direction.

If while jogging an axis, the key is pressed, the axis will move at a feedrate established by machine parameter "G00FEED" for this axis as long as this key stays pressed. When releasing this key, the axis will recover the previous feedrate (with its override %).

5.1.3 JOGGING WITH ELECTRONIC HANDWHEEL

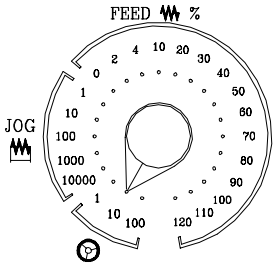
With this CNC, it is possible to govern a machine with one or more electronic handwheels.

5.1.3.1 THE MACHINE HAS ONE ELECTRONIC HANDWHEEL

Set the MFO switch to one of the handwheel positions

Positions 1, 10 and 100, indicate the multiplying factor applied to the pulses supplied by the electronic handwheel.

For example, if the manufacturer has set an equivalence of 0.100mm/turn for position 1 or 0.0100 inch/turn, the result will be:



Switch position	Distance traveled per turn
1	0.100 mm or 0.0100 inch
10	1.000 mm or 0.1000 inch
100	10.000 mm or 1.0000 inch

Select the axis to be jogged

Press one of the JOG keys of the axis to be jogged. The selected axis will appear highlighted.

When using a Fagor handwheel with axis selector button, the axis to be moved can be selected as follows:

By pressing the push-button on the back of the handwheel. The CNC selects the first axis and it highlights it.

By pressing the button again, the CNC selects the next axis and so on going from the last axis back to the first one.

By holding the button pressed for more than 2 seconds, the CNC will de-select that axis.

Move the axis

Once the axis has been selected, the machine will move it as the handwheel is being turned while respecting the turning direction applied to it.

Warning:

It could happen that depending on how fast the handwheel is turned and on the switch position, the CNC may be demanded to move the axes faster than the maximum feedrate allowed by axis machine parameter "G00FEED". In that case, the CNC will move the axis the distance indicated, but it will limit its feedrate to that maximum value.

5.1.3.2 THE MACHINE HAS SEVERAL HANDWHEELS

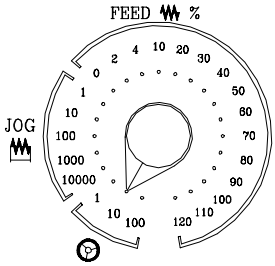
When the machine has several electronic handhweels, each one is associated with a specific axis (up to 3 handwheels).

Place the MFO switch to one of the handwheel positions

Positions 1, 10 and 100, indicate the multiplying factor applied to the pulses supplied by the electronic handwheel.

For example, if the manufacturer has set an equivalence of 0.100mm/turn for position 1 or 0.0100 inch/turn, the result will be:

Switch position	Distance traveled per turn
1	0.100 mm or 0.0100 inch
10	1.000 mm or 0.1000 inch
100	10.000 mm or 1.0000 inch



Move the axis

La máquina desplazará cada uno de los ejes según se vaya girando el volante correspondiente, teniendo en cuenta la posición seleccionada en el conmutador y respetándose además el sentido de giro aplicado.

Warning:

It could happen that depending on how fast the handwheel is turned and on the switch position, the CNC may be demanded to move the axes faster than the maximum feedrate allowed by axis machine parameter "G00FEED". In that case, the CNC will move the axis the distance indicated, but it will limit its feedrate to that maximum value.

5.2 MANUAL CONTROL OF THE SPINDLE

It is possible to control the spindle by means of the following Operator-Panel keys without the need to execute M03, M04 or M05.

is similar to executing M03. It starts the spindle clockwise and it displays M03 in the history of machining conditions.

is similar to Executing M04. It starts the spindle counter-clockwise and it displays M04 in the history of machining conditions.

is similar to executing M05. It stops the spindle.

and

vary the programmed spindle speed between the % set in spindle machine parameters “MINSOVR” and “MAXSOVR with incremental steps set in spindle machine parameter “SOVRSTEP”.

It is recommended to define the spindle speed before selecting the turning direction in order to avoid an abrupt start.

6. *TABLES*

In order to select a new tool, tool offset or zero offset, it is necessary that those values be previously stored at the CNC.

The tables available at the CNC are:

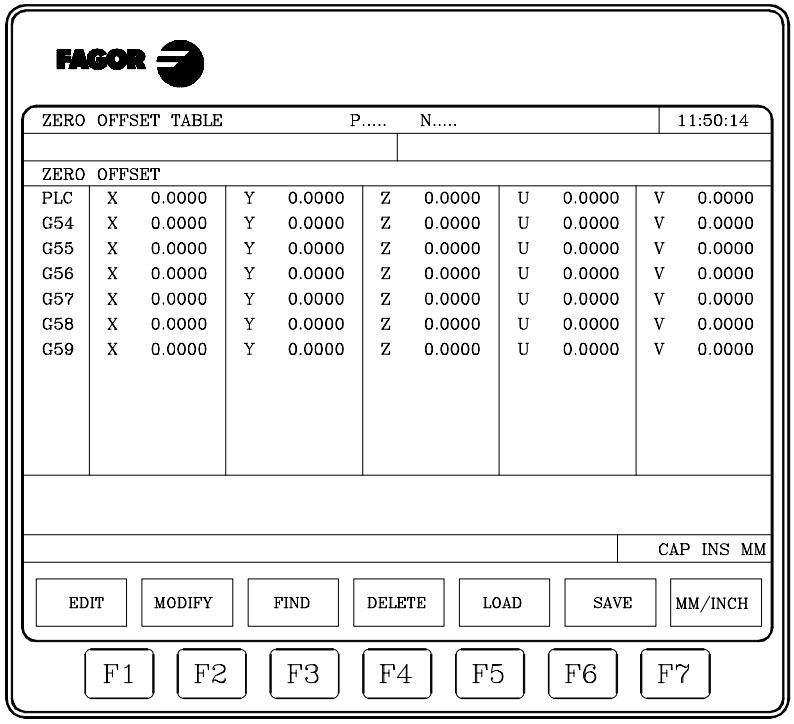
* Zero offset table
* Tool offset table
* Tool table
* Tool magazine table
* Global and local parameter table

It is recommended to save the tables in the "Memkey Card" or out to a peripheral device or PC.

When accessing the TABLES operating mode, the CNC shows all the tables saved into the "Memkey Card" (CARD A)

6.1 ZERO OFFSET TABLE

This table stores the offset of each axis.



The possible zero offsets are

Additive zero offset defined by PLC.

It is used, among others, to compensate for possible deviations due to machine dilatation.

These values are set from the PLC and from the part-program, by means of high level variables "PLCOF(X-C)".

The CNC always adds these values to the zero offset currently active.

Absolute zero offsets G54 through G57.

How to edit these tables is described later on. They can also be modified from the PLC and from the part-program, by means of high level variables "ORG(X-C)".

In order for one of these absolute zero offsets to be active, it must be selected at the CNC using its corresponding G code (G54, G55, G56 or G57).

Incremental zero offsets G58 and G59.

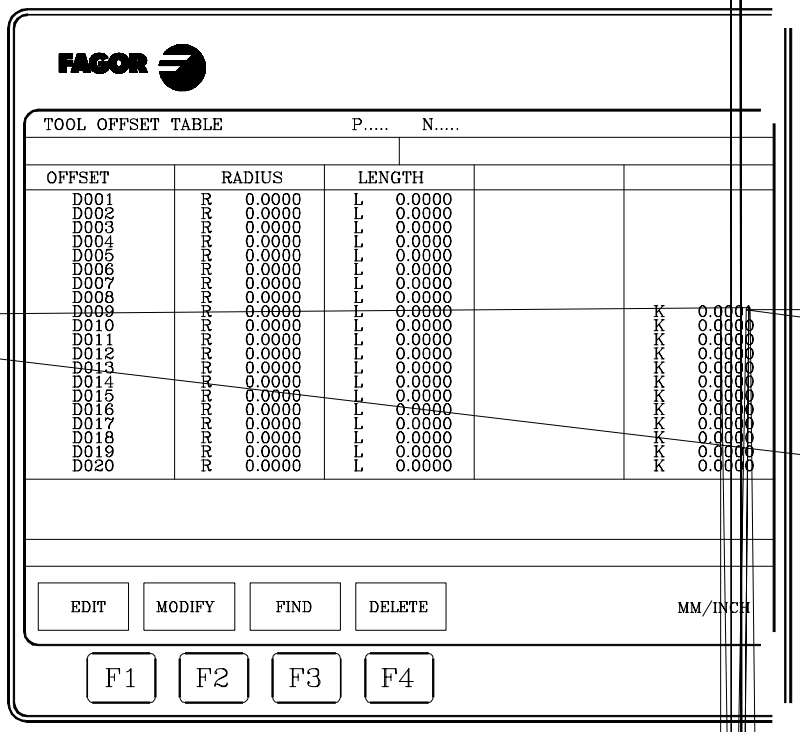
How to edit these tables is described later on. They can also be modified from the PLC and from the part-program, by means of high level variables "ORG(X-C)".

In order for one of these incremental zero offsets to be active, it must be selected at the CNC using its corresponding G code (G58 or G59).

The new incremental zero offset will be added to the absolute zero currently selected.

6.2 TOOL OFFSET TABLE

This table stores the dimensions of each tool.



Each offset has a number of fields containing the tool dimensions. These fields are:

Tool radius.

Tool length.

Tool radius wear

The CNC will add this value to the nominal radius to calculate the real tool radius (R+I).

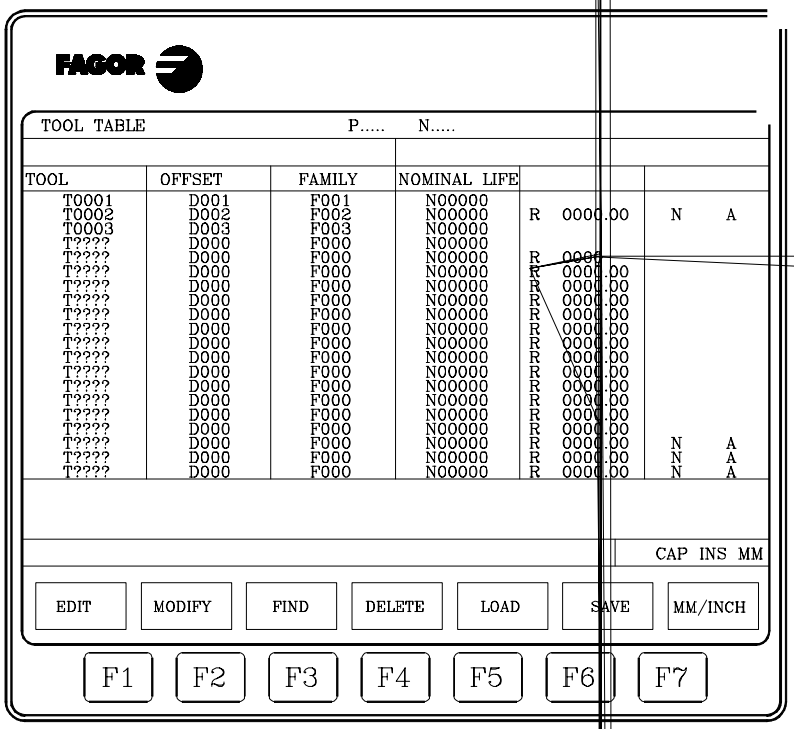
Tool length wear.

The CNC will add this value to the nominal length to calculate the real tool length (Z+K).

How to edit these values will be described later on. They can also be modified from the PLC and from the part-program by means of the high level variables associated with the tools.

6.3 TOOL TABLE

This table stores information about the tools available indicating the type of tool offset associated with them, their family, etc.


FAGOR
TOOL TABLE P..... N.....
TOOL OFFSET FAMILY NOMINAL LIFE
T0001 D001 F001 N00000
T0002 D002 F002 N00000 R 0000.00 N A
T0003 D003 F003 N00000
T???? D000 F000 N00000
CAP INS MM
EDIT MODIFY FIND DELETE LOAD SAVE MM/INCH
F1 F2 F3 F4 F5 F6 F7

Each tool has the following data fields:

Offset number associated with the tool.

Every time a tool is selected, the CNC will assume its dimensions as they appear in the tool offset table for the specified tool offset.

Family code.

It will be used when having an automatic tool changer and it will allow replacing the worn out tool with another one with similar characteristics.

There are two types of families:

* Those for normal tools whose codes are between 0 and 199.

* Those for special tools (which occupy more than one magazine pocket), whose numbers are between 200 and 255.

Every time a new tool is selected, the CNC checks whether it is worn out ("real life" greater than "nominal life"). If that is the case, it will not select it, but it will select another one of the same family, instead.

If while machining a part, the PLC "asks" the CNC to reject the current tool (by activating the logic input “TREJECT”), the CNC will display the message "rejected" in the "STATUS" field and it will replace it with the next tool of the same family that

appear in the tool table. This change will take place the next time that tool is selected.

Nominal tool life.

It indicates the machining time (in minutes) or the number of operations that that tool is calculated to last.

Real tool life.

It indicates the machining time (in minutes) or the number of operations already carried out by that tool.

Tool status.

It indicates the size of the tool and its status:

The tool size depends on the number of pockets it takes in the magazine and it is defined as follows:

N = Normal (family 0-199)
S = Special (family 200-255)

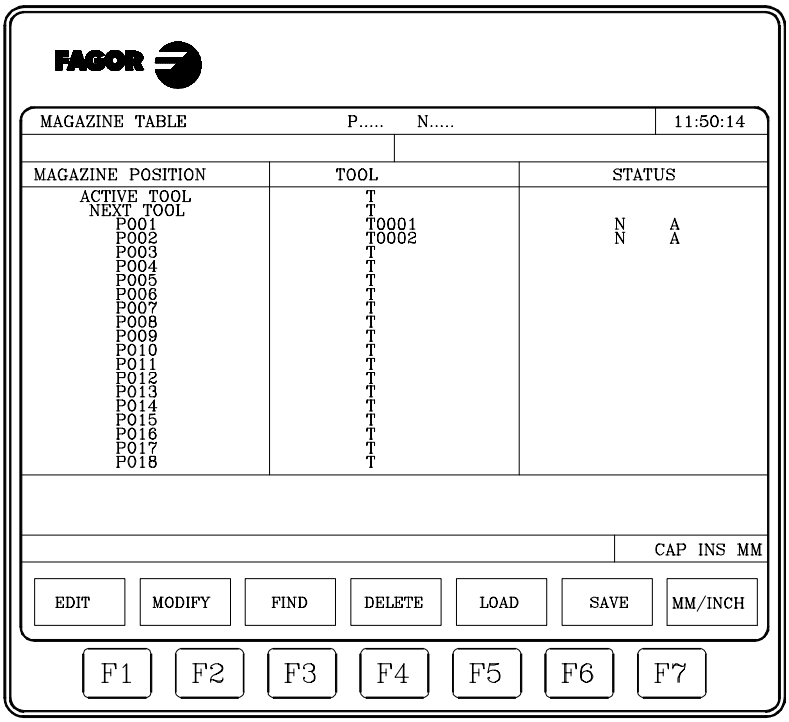
The tool status is defined as follows:

A = Available
E = Expired ("real life" greater than "nominal life")
R = Rejected by the PLC

How to edit these values is described later on.

6.4 TOOL MAGAZINE TABLE

This table contains information about the tool magazine indicating all the tools of the magazine and their position in it.



Magazine position

Besides indicating each position in the magazine, it indicates the active tool and the one selected for the following operations.
The next tool will be placed in the spindle after executing auxiliary function M06.

Tool

It indicates the number of the tool occupying that position (pocket).
The empty pockets appear with the letter "T" and the canceled ones with the characters T****.

Status

The first letter indicates the tool size and the second one its status.

The size depends on the number of pockets it occupies in the magazine.

N = Normal (family 0-199) S = Special (family 200-255)

The tool status is defined as following:

A = Available
E = Expired (“real life” greater than “nominal life”)
R = Rejected by the PLC

How to edit these values is described later on.

6.5 GLOBAL AND LOCAL PARAMETER TABLES

The CNC has two types of general purpose variables:

Local parameters P0-P25 (7 levels)
Global parameters P100-P299.

The CNC updates the parameter tables after carrying out the operations indicated in the block in preparation. This operation is always carried out before executing the block. Therefore, the values shown in the table may not coincide with those of the block being executed.

When quitting the Execution mode after interrupting program execution, the CNC updates the parameter tables with the values corresponding to the block that was being executed.

In the global and local parameter tables, the values of the parameters may be displayed in decimal (4127.423) or in scientific notation (0.23476 E-3).

The CNC generates a new nesting level of local parameters every time parameters are assigned to a subroutine. Up to a maximum of 6 nesting levels of local parameters are possible.

Machining canned cycles G66, G68, G69, G81, G82, G83, G84, G85, G86, G87, G88 and G89 use the sixth nesting level of local parameters when they are active.

To access the different local parameter tables, the corresponding level must be indicated (0 through 6).

While programming in high level, local parameters may be referred to as P0-P25, or as **A-Z**,"A" being the same as "P0" and "Z" the same as "P25".

To do that, the local parameter tables show the letter associated to them, in brackets, next to the parameter number. In the tables, the parameter can only referred to as P0-P25, letters are not allowed.

6.6 HOW TO EDIT TABLES

The screen may be scrolled up and down line by line using the up/down arrow keys or page by page using the page up/down keys.

There are several ways to edit or modify a line which will be described next.

Once the user has selected any of those options, a editing area is available on the screen which may be scrolled up and down using the up/down arrow keys. On the other hand, with the up arrow key, the cursor may be placed over the first character of the editing window and, with the down arrow key over the last character.

EDIT

Once this option has been selected, the softkeys change color appearing over a white background and they show the information corresponding to the type of editing that may be done.

On the other hand, more information on the editing commands can be obtained at any time by pressing [HELP]. To quit this help mode, press [HELP] again.

Press [ESC] to quit the editing mode and maintain the table with the previous values.

Once the editing is done, press [ENTER]. The values assigned will be entered into the table.

MODIFY

Once this option has been selected, the softkeys change color appearing over a white background and showing the information corresponding to each field.

On the other hand, more information on the editing commands can be obtained at any time by pressing [HELP]. To quit this help mode, press [HELP] again.

By pressing [ESC], the information shown in the editing area is deleted. From this point on, the selected line may be edited again.

To quit the "modify" option, the information appearing in the editing area must be deleted by pressing [CL] or [ESC] and then [ESC]. The table will keep its previous values.

Once the modification is over, press [ENTER]. The new values assigned will be entered into the table.

FIND

Once this option has been selected, the softkeys will show the following options:

BEGINNING When pressing this softkey, the cursor is placed over the first line of the table that can be edited.

END When pressing this softkey, the cursor is placed over the last line of the table.

ZERO OFFSET, TOOL OFFSET, TOOL, POSITION, PARAMETER
When pressing one of these softkeys, the CNC requests the field number to be found. Once the field has been defined, press [ENTER].

The CNC searches for the requested field and places the cursor over it (when found).

DELETE

When deleting a line, the CNC sets all the fields to "0".

To delete a line, indicate its number and press [ENTER]

To delete several lines, indicate the beginning, press the [UP TO] softkey, indicate the last line to be deleted and press [ENTER]

To delete all the lines, press the "ALL" softkey. The CNC requests confirmation of the command.

INITIALIZE

It deletes all the data of the table by setting them all to "0". The CNC requests confirmation of the command.

LOAD

The tables may be loaded from the "Memkey Card" (CARD A) or a peripheral device or a PC through the two serial communications lines (RS232C or RS422).

The transmission starts after pressing the corresponding softkey. When using a serial line, the receptor must be ready before starting the transmission.

To interrupt the transmission, press the "ABORT" softkey.

If the length of the table received does not coincide with the current table length, the CNC will act as follows:

If the table received is shorter than the current one, the received lines are modified and the rest remain with their previous values.

If the table received is longer than the current one, all the lines of the table are modified and when detecting that there is no more room, the CNC will issue the corresponding error message

SAVE

The tables may be saved into the "Memkey Card" (CARD A) or out to a peripheral device or PC through the two serial lines (RS232C or RS422).

The transmission starts after pressing the corresponding softkey. When using a serial line, the receptor must be ready before starting the transmission.

To interrupt the transmission, press the "ABORT" softkey.

MM/INCHES

It toggles the display units for the data. The lower right-hand side window shows the units selected (MM/INCH).

7. UTILITIES

In this operating mode, one can access the programs stored in the CNC's RAM memory, in the "Memkey Card" (CARD A), in the hard disk (HD) and in external devices (through the serial lines 1 and 2).

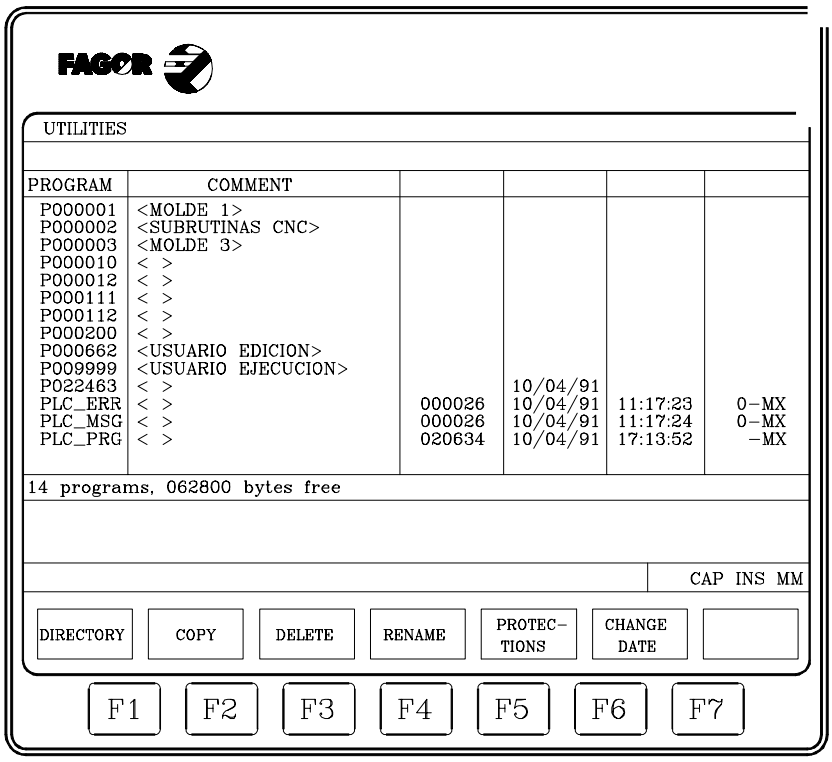
They can be deleted, renamed or their protection changed. It is also possible to make copies within the same device or from one to another.

7.1 DIRECTORY

To access the program directory of the CNC's RAM memory, the "Memkey Card" (CARD A), the hard disk (HD) and of the external devices (through serial lines 1 and 2).

The subroutine directory of the CNC can also be accessed.

Program directory.
By default, the CNC shows the program directory of the RAM memory, to view another directory, press the corresponding softkey.



On each directory, the CNC shows all the programs visible (not hidden) to the user, that is:

Part programs
Customizing programs
The PLC program (PLC_PRG)
The PLC error file (PLC_ERR)
The PLC message file (PLC_MSG)

The program directory has the following definition fields:

Program
It shows the number when it is a part-program or a customizing program and the corresponding mnemonic when it is a PLC program, the PLC error file or the PLC message file.

Comment
Any program may have a comment associated with it for its identification.
The comments may be defined when editing the program or in this operating mode using the Rename option as described later on.

Size
It indicates, in bytes, the size of the program text. It must be borne in mind that the actual size of the program is slightly greater because this field does not include the space occupied by some variables used internally (header, etc.).

The date and the time when the program was edited (last changed)

Attributes
They show information about the source and usefulness of each program. The attributes are defined in this operating mode by means of the Protections option as described later on.

***** The program is running, either because it is the main program or because it contains a subroutine which has been called upon from that program or from another subroutine.

O The program was created by the machine manufacturer.

H The program is hidden and cannot be displayed in any directory.

Since a hidden program can nevertheless be edited or deleted if its number is known, it is recommended to remove the "Modifiable" attribute to prevent it from being edited or deleted.

M The program may be modified. In other words, it may be edited, copied, etc.

If a program does not have this attribute, the operator cannot see or modify its contents.

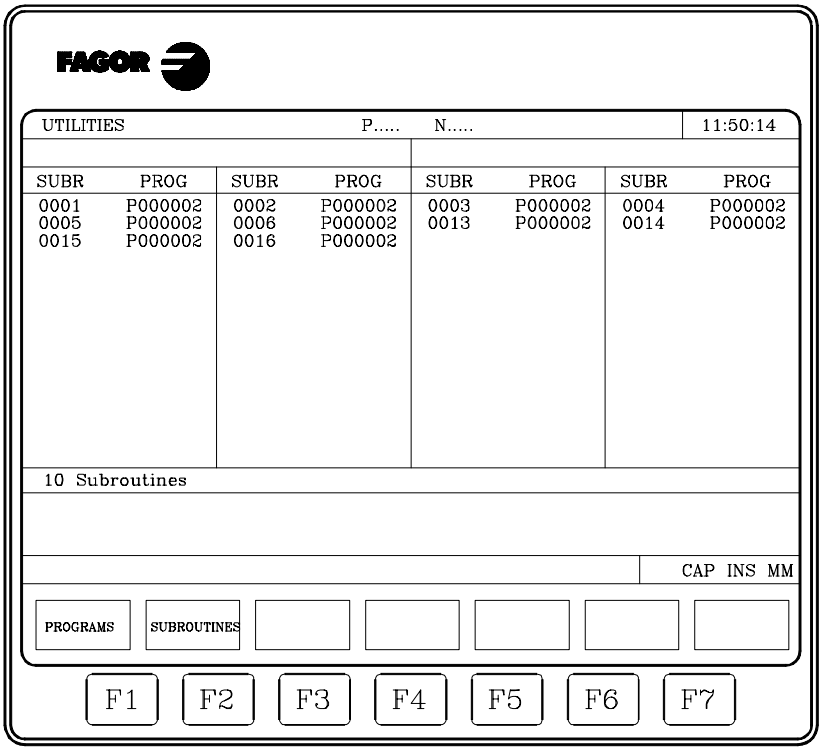
X Indicates that the program may be executed.

A program not having this attribute cannot be executed by the operator.

Only the attributes currently selected will be shown, the ones not selected will appear as "-".

Example: O—X Indicates that the program was created by the manufacturer, it will be displayed in the directory (not hidden), it cannot be modified, but it may be executed.

Subroutines directory.



It lists all the subroutines defined in the part programs of the CNC ordered from the smallest one to the largest one.

Also, next to the subroutines, it displays the number of the program where it has been defined.

If the program containing the subroutine has the "hidden" attribute assigned to it, that program number will appear as P??????.

7.1.1 DIRECTORY OF THE EXTERNAL DEVICES

When accessing the directory of an external device through the serial lines, that directory is shown in DOS format.

The [CHANGE DIR] softkey lets the user select the work directory of the PC to operate with from the CNC.

This operation does not change the work directory that was selected to operate with from the PC.

In other words, when working via DNC, it is possible to select a work directory at the PC and another PC directory at the CNC.

This new feature is available from DNC50 version 5.1 on.

7.2 COPY

To copy programs in the same directory or between directories of different devices.

The copies may be made between:
The CNC's RAM memory, "Memkey Card" (CARD A), hard disk (HD) and external devices (serial lines)

To make a copy, proceed as follows:

Press the [COPY] softkey
Indicate the location of the program to be copied (RAM memory, CARD A, HD or DNC)
key in the program number to be copied
Press the [IN] softkey
Indicate the destination of the copy (RAM memory, CARD A, HD or DNC)
Key in the program number
Press [ENTER]

Example to copy program 200103, from the CNC's RAM memory out to the "Memkey Card" with the number 14

COPY (MEMORY) P200103 IN (CARD A) P14 ENTER

If a program with the same number already exists, the CNC will display a warning message. On the other hand, if that program is in execution, the CNC will display a message indicating that it is not possible.

Two subroutines may not have the same name in RAM memory. To make a copy and change the name of the copied subroutine, write the subroutine defining block as a comment **before** making the copy.

7.3 DELETE

A program may be deleted from the CNC' RAM memory, from the "Memkey Card" (CARD A), from the Hard Disk (HD) or from the external devices (through the serial lines 1 and 2).

To delete a program, proceed as follows:

Press the [DELETE] softkey
Indicate the location of the program to be deleted (RAM, CARD A, HD or DNC).
Key in the number of the program to be deleted or place the cursor over it.
Press [ENTER]

Example to delete program 200103 from the "Memkey Card"

DELETE (CARD A) P200103 ENTER

Only programs that can be modified ("M" attribute) can be deleted.

7.4 RENAME

To rename or assign a new comment to a program stored in the CNC's RAM memory, "Memkey Card" (CARD A), or in the Hard Disk (HD).

To rename a program, proceed as follows:

Press the [RENAME] softkey.
Indicate the location of the program (RAM, CARD A, HD or DNC).
Key in the number of the program to be renamed
Press the [TO] softkey.
Press either the [NEW NUMBER] or the [NEW COMMENT] softkey
Key in the new number or the new comment
Press [ENTER].

The files associated with the PLC (program, messages and errors) are always referred to with their associated mnemonics. Therefore, only their comment may be renamed.

If there is a program with the same number, the CNC will issue a warning message and it will offer the chance to modify the command.

Examples:

To change the name of program 200103 from the "Memkey Card"

RENAME (CARD A) P200103 TO NEW NUMBER P12 ENTER

to change the comment of program 100453 from the CNC

RENAME (MEMORY) P100453 TO NEW COMMENT "Test" ENTER

7.5 PROTECTIONS

To prevent certain programs from being manipulated and restrict access to the operator to certain CNC commands.

It is possible to protect programs stored in the CNC's RAM memory, in the "Memkey card" (CARD A) or in the Hard Disk (HD).

USER PERMISSIONS

Lets the operator see those CNC programs that have been created by the operator and sets their attributes.

To modify the attributes of a program, proceed as follows:
Press the [USER PERMISSION] softkey
Indicate the location of the program (RAM MEMORY, CARD A or HD)
Key in the number of the program whose attribute is to be changed
Press the softkeys

F2	to change the (H) attribute	hidden/visible program
F3	to change the (M) attribute	modifiable program
F4	to change the (X) attribute	executable program

Press [ENTER]

OEM PERMISSION

Lets see all the programs stored at the CNC whether they are created by the OEM or by the operator and set their attributes.

To modify the attributes of a program, proceed as follows:
Press the [OEM PERMISSION] softkey
Indicate the location of the program (RAM MEMORY, CARD A or HD)
Key in the number of the program whose attribute is to be changed
Press the softkeys

F1	to change the (O) attribute	OEM program
F2	to change the (H) attribute	hidden/visible program
F3	to change the (M) attribute	modifiable program
F4	to change the (X) attribute	executable program

Press [ENTER]

PASSWORDS

Lets define each of the passwords that the operator must key in before accessing the various CNC commands.

General access password (MASTERPSW)
It is requested when trying to access this password option.
(Utilities mode / Protections / Passwords).

OEM password (OEMPSW)
It is requested when trying to access OEM permissions
(Utilities mode / Protections / OEM permissions).

User password (USERPSW)
It is requested when trying to access user permissions
(Utilities mode / Protections / User permissions).

PLC access password (PLCPSW)

It is requested in the following cases:

- When editing the PLC program, the PLC message program and the PLC error program.
- When compiling the PLC program.
- When trying to change the status of a resource or execute a program execution controlling command.

Customizing password (CUSTOMPSW)

It is requested when trying to access the Customizing mode.

Machine parameter access password (SETUPPSW)

It is requested when trying to access the options to modify the table values (Edit, Modify, Initialize, Delete and Load) except for tables of the serial lines which are not protected.

To change or delete the passwords, use the following softkeys:

Change password.

Select the desired password and enter the new one.

Delete password.

Lets delete (eliminate) one of several codes from the table.

- To delete a password, indicate its number and press [ENTER].
- To delete several passwords (they must be in a row), indicate the number of the first one to be delete, press the "UPTO" softkey, indicate the number of the last one to be deleted and press [ENTER].
- To delete a password, indicate its number and press [ENTER].

Clear all.

Lets delete all the passwords. The CNC will request confirmation of the command and it will delete them after pressing [ENTER].

7.6 CHANGE DATE

Lets change the system date and time.

First, the date will be shown as day/month/year (12/04/1998). After changing it, press [ENTER] to validate it. If it is not to be changed, press [ESC].

Next, the time will be shown as hours/minutes/seconds (08/30/00). After changing it, press [ENTER] to validate it. If it is not to be changed, press [ESC].

8. DNC

This operating mode shows the CNC status after a power outage and the status of the DNC communication lines. It also lets activate and deactivate DNC communications with a PC.

8.1 CNC

This screen shows the number of the line that was being executed last time an execution error or a power outage occurred.

The CNC shows the program number and line number that was executing as well as where the program is stored.

On the other hand, if that program called upon a subroutine and the CNC was executing it, it will display:

The subroutine number, the program containing its definition and the line or block of the subroutine being executed.

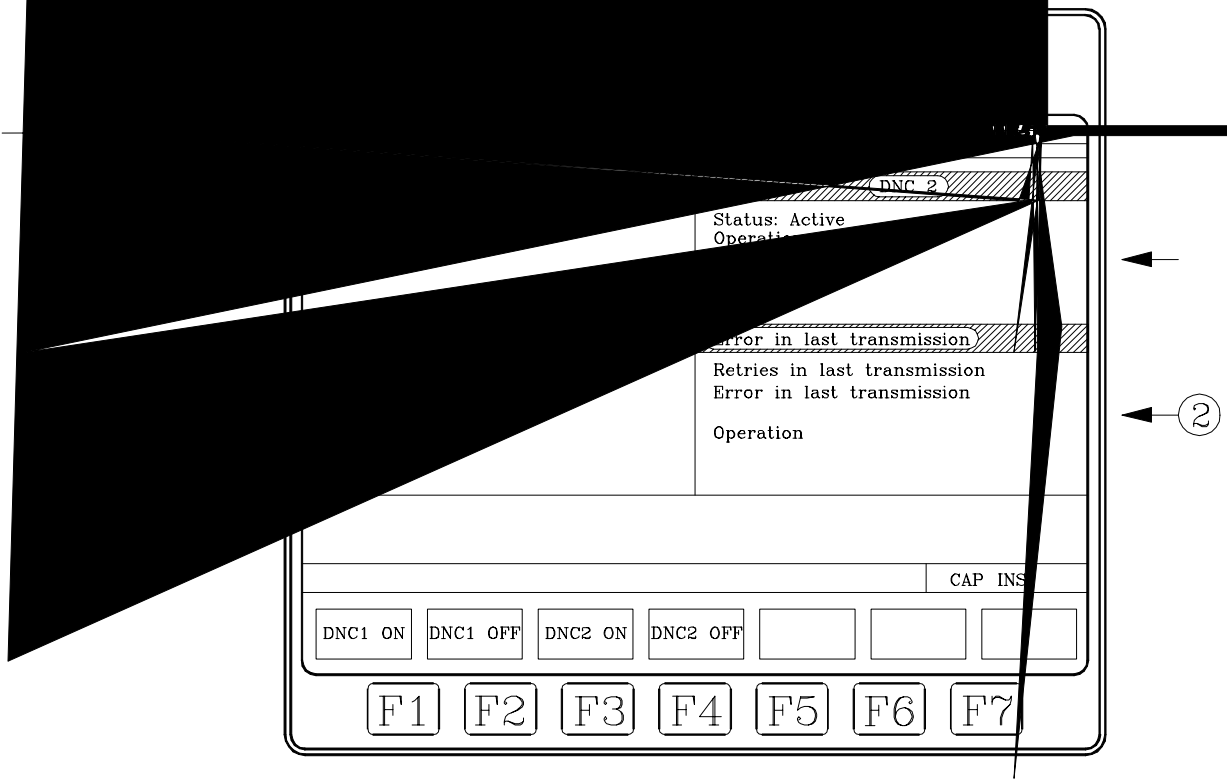
Example:

Device	Program	Line number	Subroutine
CARD A	000012	7	
MEMORY	001000	15	0033

Indicates that the CNC was executing line 7 of program 12 of CARD A.

That program line called to subroutine 15 and it was executing its line number 33. That subroutine is defined (contained) in program 1000 which is stored in the CNC's RAM memory.

least one of the serial
communicate with the



The left-hand side of the screen corresponds to serial line 1 and the right-hand side to serial line 2.

In the example of the figure above, serial line 1 is used to communicate with a Fagor Floppy Disk Unit; and serial line 2 to communicate via DNC.

The upper area, 1, indicates:

* The status of the serial line: Active / Inactive.

* The type of operation in progress:
  Sending program / Receiving program / Sending directory /
  Receiving directory / etc.

The lower area, 2, indicates the last operation and the type of error occurred if any.

9. *PLC*

In this mode of operation it is possible to access the PLC to check its operation or the status of the various PLC variables. It also allows editing and analyzing the PLC program as well as the PLC message file and error file.

The accessible programs associated with the PLC are:

The PLC program (PLC_PRG)
The PLC error file (PLC_ERR)
The PLC message file (PLC_MSG)

The PLC program (PLC_PRG) may be edited at the front panel or copied from the "Memkey Card" (CARD A) or from a peripheral device or PC.

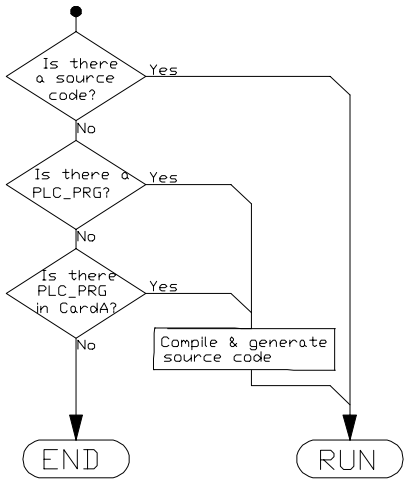
The PLC program (PLC_PRG) is stored in the internal CNC memory with the part-programs and it is displayed in the program directory (utilities) together with the part-programs.

Before executing the PLC_PRG program, it must be compiled.
Once it is done compiling, the CNC requests whether the PLC should be started or not.

To make the operator life easier and avoid new compilations, the source code generated at each compilation is stored in memory.

After power-up, the CNC acts as follows:

- Runs the executable program stored in memory.
- If there isn't one, it compiles the PLC_PRG program already in memory and runs the resulting executable program.
- If there isn't one, it looks for it in the "Memkey Card" (CARD A)
- If it isn't in the CARD A either, it does nothing. Later on, when accessing the Jog mode, Execution mode, etc. the CNC will issue the corresponding error message.



Once the program has been compiled, it is not necessary to keep the source program (PLC_PRG) in memory because the PLC always executes the executable program.

Once the proper performance of the PLC has been verified, it is a good idea to save it into the "Memkey Card" (CARD A) using the instruction SAVE PROGRAM (as described later on).

9.1 EDIT

Once this option is selected, indicate with the corresponding softkey the PLC program to be edited.

The PLC program (PLC_PRG)
The PLC error file (PLC_ERR)
The PLC message file (PLC_MSG)

The cursor can be moved line by line with the “up and down” arrow keys or page by page with the “page up” and “page down” keys.

The cursor position or line number will be displayed in a white window inside the communications window (bottom of the screen) next to the CAP/INS indicator window.

This operating mode offers various options which are described next.

Once any of these functions is selected, the CNC shows an editing area on the CRT where the cursor may be moved by using the up/down and right/left arrow keys. Also, the up-arrow key positions the cursor over the first character of the editing area and the down-arrow key positions the cursor over the last character.

EDIT

With this option it is possible to edit new lines or blocks of the selected program.

Before pressing this softkey, the block after which the new ones will be added must be selected with the cursor.

The program will be edited (written) a block at a time and each block can be written in ISO language, High Level language or it can be just a program comment.

Once this option is selected, the softkeys will change their color showing their type of editing option over a white background.

Also, it is possible to get more editing assistance by pressing **HELP**. Press **HELP** again to exit the editing assistance mode.

Press the **ESC** key to exit the block editing mode when writing a block and this block will not be added to the program.

Once the block has been edited, press **ENTER** to add it to the program behind the block previously indicated by the cursor.

The cursor will be positioned at the new block (just edited) and the editing window (area) will be cleared In order to edit a new block.

Press **ESC** or **MAIN MENU** to quit the block editing mode.

MODIFY

This option permits modifying the contents of a selected program block.

Before pressing this softkey, select with the cursor the block to be modified .

Once this option is selected, the softkeys will change their color showing their type of modifying option over a white background.

Also, it is possible to get more editing assistance by pressing **HELP**. Press **HELP** again to exit the editing assistance mode.

By pressing **ESC**, the information corresponding to that block and which was shown in the editing area will be cleared. It will then be possible to modify its contents again.

To quit the block modifying mode, press **CL** or **ESC** to clear the editing window and then press **ESC** again. This way, the selected block will not be modified.

Once the block contents have been modified, press **ENTER** so the new contents replace the old ones.

FIND

This option is used to find a specific text within the selected program.

When selecting this option, the following options will appear:

BEGINNING This softkey positions the cursor over the first program block which is then selected quitting the "find" option.

END This softkey positions the cursor over the last program block which is then selected quitting the "find" option.

TEXT With this function it is possible to search a text or character sequence starting from the block indicated by the cursor.

When this key is selected, the CNC requests the character sequence to be found.

When the text is defined, press the "END OF TEXT" softkey and the cursor will be positioned over the first occurrence of that text.

The search will begin at the current block.

The text found will be highlighted being possible to continue with the search or to quit it.

Press **ENTER** to continue the search up to the end of the program. It is possible to search as many times as wished and when the end of the program is reached, it will start from the first block.

Press the "EXIT" softkey or the **ESC** key to quit the search mode. The cursor will be positioned where the indicated text was found last.

LINE NUMBER After pressing this key, the CNC requests the number of the block to be found. After keying in the desired number and pressing **ENTER**, the cursor will position over that block which will then be selected quitting the search mode.

REPLACE

With this function it is possible to replace a character sequence with another throughout the selected program.

When selecting this option, the CNC requests the character sequence to be replaced.

Once the text to be replaced is indicated, press the "WITH" softkey and the CNC will request the character sequence which will replace the previous one.

Once this text is keyed in, press the "END OF TEXT" softkey and the cursor will be positioned over the first occurrence of the searched text.

The search will begin at the current block.

The found text will be highlighted and the following softkey options will appear:

REPLACE Will replace the highlighted text and will continue the search from this point to the end of the program.

If no more occurrences of the text to be replaced are found, the CNC will quit this mode.

If another occurrence of the text is found, it will be highlighted showing the same "replacing" or "not replacing" options.

DO NOT REPLACE Will not replace the highlighted text and will continue the search from this point to the end of the program.

If no more occurrences of the text to be replaced are found, the CNC will quit this mode.

If another occurrence of the text is found, it will be highlighted showing the same "replacing" or "not replacing" options.

TO THE END This function will automatically replace all the matching text from the current block to the end of the program without offering the option of not replacing it.

ABORT This function will not replace the highlighted text and it will quit the "find and replace" mode.

DELETE BLOCK

With this function it is possible to delete a block or group of blocks.

To delete only one block, just position the cursor over it and press **ENTER**.

To delete a group of blocks, indicate the first and last blocks to be deleted. To do so, follow these steps:

* Position the cursor over the first block to be deleted and press the "INITIAL BLOCK" softkey.

* Position the cursor over the last block to be deleted and press the "FINAL BLOCK" softkey.

  If the last block to be deleted is also the last one of the program, it can also be selected by pressing the "TO THE END" softkey.

* Once the first and last blocks are selected, the CNC will highlight the selected blocks requesting confirmation to delete them.

MOVE BLOCK

With this option it is possible to move a block or group of blocks by previously indicating the first and last blocks to be moved. To do so, follow these steps:

* Position the cursor over the first block to be moved and press the "INITIAL BLOCK" softkey.

* Position the cursor over the last block to be moved and press the "FINAL BLOCK" softkey.

  If the last block to be moved is also the last one of the program, it can also be selected by pressing the "TO THE END" softkey.

  To move only one block, the "initial block" and the "final block" will be the same one.

* Once the first and last blocks are selected, the CNC will highlight the selected blocks requesting confirmation to move them.

  Then, indicate the block after which this group of blocks must be placed.

* Press the "START OPERATION" softkey to carry out the move.

COPY BLOCK

With this option it is possible to copy a block or group of blocks by previously indicating the first and last blocks to be copied. To do so, follow these steps:

* Position the cursor over the first block to be copied and press the "INITIAL BLOCK" softkey.

* Position the cursor over the last block to be copied and press the "FINAL BLOCK" softkey.

  If the last block to be copied is also the last one of the program, it can also be selected by pressing the "TO THE END" softkey.

  To copy only one block, the "initial block" and the "final block" will be the same one.

* Once the first and last blocks are selected, the CNC will highlight the selected blocks requesting confirmation to copy them.

  Then, indicate the block after which this group of blocks must be placed.

* Press the "START OPERATION" softkey to carry out this command.

COPY TO PROGRAM

With this option it is possible to copy a block or group of blocks of one program into another program.

When selecting this option, the CNC will request the number of the destination program where the selected block or blocks are to be copied. After entering the program number press **ENTER**.

Next, indicate the first and last blocks to copy by following these steps:

* Position the cursor over the first block to be copied and press the "INITIAL BLOCK" softkey.

* Position the cursor over the last block to be copied and press the "FINAL BLOCK" softkey.

  If the last block to be copied is also the last one of the program, it can also be selected by pressing the "TO THE END" softkey.

  To copy only one block, the "initial block" and the "final block" will be the same one.

* Once the first and last blocks are selected, the CNC will highlight the selected blocks and will execute the command.

  If the destination program already exists, the following options will be displayed:

* Write over the existing program. All the blocks of the destination program will be erased and will be replaced by the copied blocks.

* Append (add) the copied blocks behind the ones existing at the destination program.

* Abort or cancel the command without copying the blocks.

INCLUDE PROGRAM

With this option it is possible to include or merge the contents of another program into the one currently selected.

Once this option is selected, the CNC will request the number of the source program to be merged. After keying in that number press **ENTER**.

Next, indicate with the cursor the block after which the source program will be included.

Finally, press the "START OPERATION" softkey to execute the command.

9.2 COMPILE

With this option it is possible to compile the PLC source program “PLC_PRG”.

The PLC program must be stopped in order to compile it, otherwise, the CNC will “ask” if it is desired to stop it.

Once the source program compiled, the CNC will generate the executable PLC program (object program).

If while compiling, some errors are detected, the CNC will not create the object program and the detected errors (up to 15) will appear on the screen.

If the errors do not affect the proper program execution (such as non-referenced labels, etc.), the CNC will display the corresponding warning messages but it will generate the object program.

After a successful compilation, the CNC will “ask” whether the PLC program must be started or not.

9.3 MONITORING

With this option it is possible to display the PLC program and analyze the status of the different PLC resources and variables.

Once this option has been selected, the CNC will show the source program that corresponds to the executable program (object) even when that program (source) has been deleted or modified at the CNC.

The CNC will also display all the variable consultations at logic level 1 (including those not being executed) and the actions whose conditions are met.

To display the program from a specific line on, press the "**L**" key followed by that line number and then press **ENTER**.

The operator can move the cursor around the CRT a line at a time with the up/down rowNG

The

.

TRS 1/256 = 0/1	Alters the status (0/1) of the RESET input of the indicated timer. For example: TRS2 = 0 sets the reset input of timer T2 to 0.
TGn 1/256 n = 0/1	Alters the status (0/1) of the trigger input "TGn" of the indicated timer (1 thru 256) assigning the desired time constant (n) to it. For example: TG1 22 1000 sets the trigger input 1 of timer T22 to one and it assigns a time constant of 1000 (10 seconds).
CUP 1/256 = 0/1	Alters the status (0/1) of the UP count input of the indicated counter. For example: CUP 33 = 0 sets the status of the UP input of counter C33 to 0.
CDW 1/256 = 0/1	Alters the status (0/1) of the DOWN count input of the indicated counter. For example: CDW 32 = 1 sets the status of the UP input of counter C32 to 1.
CEN 1/256 = 0/1	Alters the status (0/1) of the enable input of the indicated counter. For example: CEN 12 = 0, sets the enable input of counter 12 to 0.
CPR 1/256 n = 0/1	Alters the status (0/1) of the preset input of the indicated counter (1 thru 256). The counter will be preset with the value "n" if an up flank is produced with this instruction. For example: CPR 10 1000 =1 sets the preset input of counter C10 to 1 and also, if an up flank has occurred (being previously set to 0), the counter will be preset with a value of 1000.
C 1/256 = n	Presets the count of the indicated counter to the "n" value. For example: C42 = 1200 sets the count of counter C42 to 1200.
B 0/31 R 1/559 = 0/1	Alters the status (0/1) of the indicated bit (0/31) of the indicated register (1/559). For example: B5 R200 = 0 sets Bit 5 of register R200 to 0.
R 1/559 = n	Assigns the "n" value to the indicated register. For example: R 303 = 1200 assigns the value of 1200 to register R303.
R 1/559.1/559 = n	Assigns the "n" value to the indicated register group. For example: R234.236 = 120 assigns the value of 120 to registers R234, R235 and R236.

It must be borne in mind that when referring to a single resource, it is possible to do it using its corresponding mnemonic.

For example: /STOP=1 is interpreted by the CNC as M5001=1

CREATE WINDOW

This CNC allows the possibility of creating windows to display the status of the various PLC resources.

These windows will be shown overlapping the PLC program and the information displayed in them will be updated dynamically.

The options "MODIFY WINDOW", "ACTIVE WINDOW" and "ACTIVATE SYMBOLS" allow the manipulation of these windows.

Every time a new window is created, the CNC will assign 2 data lines to it in order to display the status of the desired resources.

There are two types of windows which can be selected with softkeys.

WINDOW TO DISPLAY TIMERS AND REGISTERS

This window is divided into two sections, one to display Timers and the other one to display Registers.

Timer. It will show one timer per line showing the following information for each one of them:

TG	Indicates the logic status of the active trigger input.
M	Indicates the status of the timer: "S" means stopped, "T" means timing and "D" means disabled.
TEN	Indicates the logic status of the Enable input.
TRS	Indicates the logic status of the Reset input.
T	Indicates the logic status of the status output of the timer.
ET	Indicates the elapsed time.
TO	Indicates the remaining time.

Key in the command **T 1/256** or **T 1/256.1/256** to request the data on a timer or group of timers and then press **ENTER**.

Register. It will display one register per line showing the following information fields for each of them:

HEX	Indicates the hexadecimal value of its contents.
DEC	Indicates the decimal value of its contents (with sign).

Key in **R 1/559** or **R 1/559.1/559** to request information on one or more registers and, then, press **ENTER**.

WINDOW TO DISPLAY COUNTERS AND BINARY DATA

This window is divided into two sections, one to display Counters and the other one to display Binary Data.

Counter. It will display one counter per line showing the following information fields for each of them:

CEN	Indicates the logic status of the ENABLE input.
CUP	Indicates the logic status of the UP COUNT input.
CDW	Indicates the logic status of the DOWN COUNT input.
CPR	Indicates the logic status of the PRESET input.
S	Indicates the status of the counter. "1" when its internal count is 0 and 0 for all other cases.
C	Indicates its count value.

Key in **C 1/256** or **C 1/256.1/256** to request information on one or more counter and, then, press **ENTER**.

Binary Data. It will show one data line per resource or group of resources requested.

The instructions available to request information of the various resources are:

I 1/256 or I 1/256.1/256	It shows the status of the selected input or group of inputs.
O 1/256 or O 1/256.1/256	It shows the status of the selected output or group of outputs.
M 1/5957 or M 1/5957.1/5957	It shows the status of the selected mark or group of marks.
B 0/31 R 1/559	It shows the status of the selected bit of the indicated register.

When requesting the status of one or more inputs, outputs or marks, the CNC will show complete data lines even when all of them have not been requested.

When using generic denominators (I / O / M) to display resources, the CNC will display 20 of them per line and 3 when using their associated mnemonics (symbols). In the latter case, the generic denomination will be displayed when no mnemonic is associated to a resource.

When requesting the status of a register bit, the CNC will display only the requested bit on the corresponding line.

MODIFY WINDOW

With this option it is possible to manipulate the active window (the one selected) by enlarging it, reducing it, clearing it or even eliminating (closing) it.

To do so, the following softkey options are available:

ENLARGE	To enlarge the size of the window by one line every time this softkey is pressed.
REDUCE	To reduce the size of the window by one line every time this softkey is pressed (minimum 2 lines).
CLEAR	To clear the contents of the active window.
CLOSE	To close the active window, the CNC will no longer display it.

ACTIVE WINDOW

With this option it is possible to select between the PLC program and each one of the windows being displayed (timers, registers, counters and binary data) in order to operate with it.

Bear in mind that the operator can only operate with the active window.

Once the active window has been selected, it Will be possible to:

Move the cursor (if the PLC program is the one active) or shift the display area with the up and down arrow keys.

Execute any command of the “MODIFY WINDOW” option.

FIND

This option will be executed regardless of which is the active window and it offers the following searching options:

BEGINNING This softkey positions the cursor over the first program block which is then selected quitting the "find" option.

END This softkey positions the cursor over the last program block which is then selected quitting the "find" option.

TEXT With this function it is possible to search a text or character sequence starting from the block indicated by the cursor.

When this key is selected, the CNC requests the character sequence to be found.

The CNC will consider a text found when it is isolated by blank spaces or separators. Thus, When looking for "I1" it will not find or stop at "I12" or "I123", but only at "I1".

When the text is defined, Press the "END OF TEXT" softkey and the cursor will be positioned over the first occurrence of that text.

The search will begin at the current block.

The text found will be highlighted being possible to continue with the search or to quit it.

Press **ENTER** to continue the search up to the end of the program. It is possible to search as many times as wished and when the end of the program is reached, it will start from the first block.

Press the "EXIT" softkey or the **ESC** key to quit the search mode.

ACTIVATE / DEACTIVATE SYMBOLS

With this option it is possible to display in all available windows the symbols or mnemonics associated to the various resources.

The names of the resources may be displayed in two ways: using their generic names (I, O, M, T, C, R) by **deactivating** symbols or using their associated symbols by **activating** them.

When a resource has no mnemonic associated to it, it will always be displayed with its generic name.

This softkey will toggle between ACTIVATE SYMBOL and DEACTIVATE SYMBOL every time is pressed in order to show which option is available.

LOGIC ANALYZER

It is especially suited to help in the machine startup and for troubleshooting errors and critical situations in signal behavior.

START PLC

When selecting this option, the CNC will start executing the PLC program from the beginning, including the CY1 cycle.

The CNC will ignore this command when it is already executing the PLC program.

FIRST CYCLE

When selecting this option, the CNC will execute only the initial cycle of the PLC program (CY1).

The CNC will ignore this command when it is already executing the PLC program.

SINGLE CYCLE

When selecting this option, the CNC will execute the main cycle of the PLC program (PRG) only once.

The CNC will ignore this command when it is already executing the PLC program.

STOP PLC

This softkey interrupts the execution of the PLC program.

CONTINUE

This softkey resumes the execution of the PLC program.

9.3.1 MONITORING WITH THE PLC IN OPERATION AND WITH THE PLC STOPPED

It must be borne in mind that the CNC initializes all physical outputs and the PLC resources on power-up, after the key sequence SHIFT-RESET and after detecting a WATCHDOG error at the PLC.

The initialization process sets all resources to "0" except those active low. They will be set to "1".

During the monitoring of the PLC program and the various PLC resources, the CNC will always show the real values of the resources.

If the PLC is on, note that a program cycle is processed in the following way:

* The PLC updates the real input values after reading the physical inputs (from the electrical cabinet).

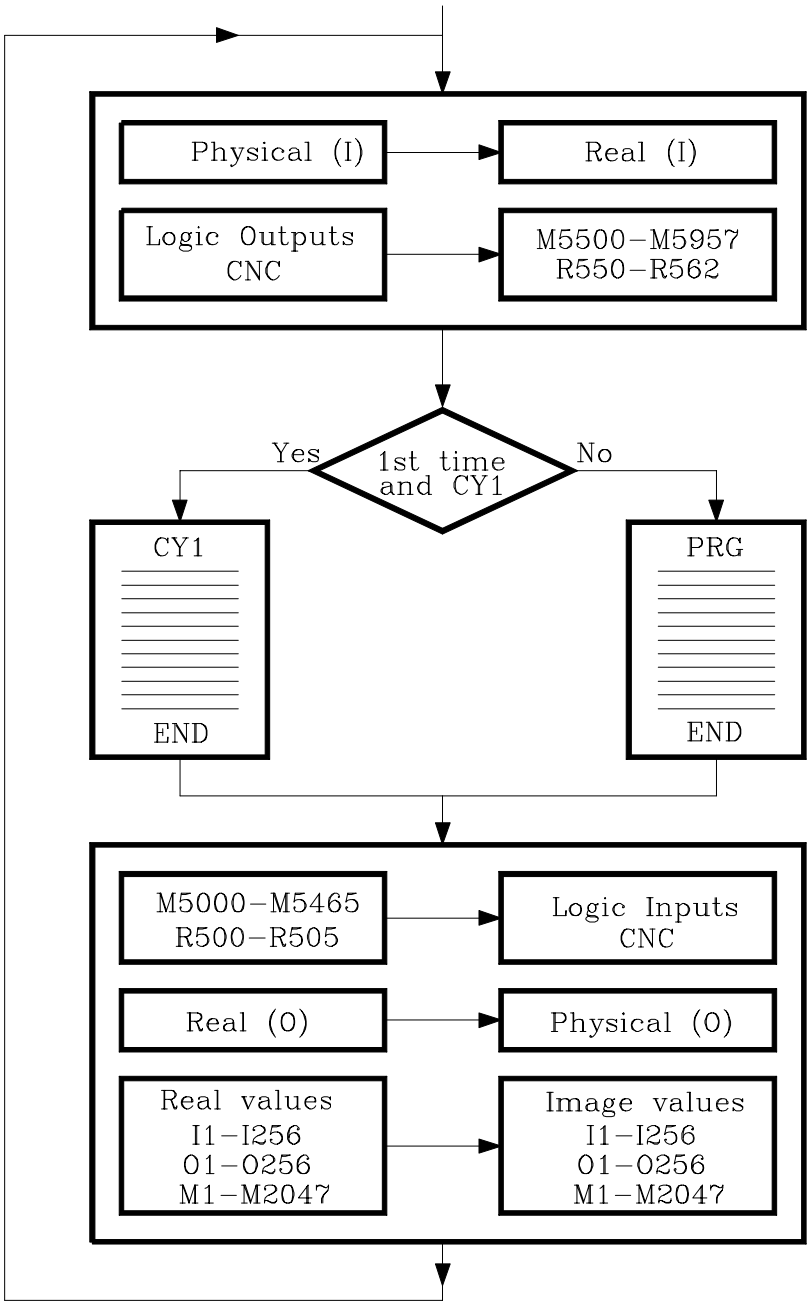
* It updates the values of resources M5000 thru M5957 and R500 thru R559 with the values of the CNC logic outputs (internal variables).

* Executes the program cycle.

* It updates the CNC logic inputs (internal variables) with the real values of resources M5000 thru M5957 and R500 thru R559.

* It assigns to the physical outputs (electrical cabinet) the real values of the corresponding "O" resources.

* It copies the real values of resources I, O, M into their own images.



If the PLC is stopped, it will work as follows:

* The real values of the "I" resources corresponding to the physical inputs will be updated every 10 milliseconds.

* The physical outputs will be updated every 10 milliseconds with the real values of the corresponding "O" resources.

* The PLC will attend to all requests and modifications of its internal variables.

9.4 ACTIVE MESSAGES

When selecting this option, the CNC will display a page (or screen) showing dynamically all the active messages generated by the PLC.

These messages will be listed by priority always starting from the one with the smallest number (highest priority).

The operator can move the cursor a line at a time with the up and down arrow keys or page by page with the page-up and page-down keys.

To delete one of the displayed messages, select it with the cursor and press the “DELETE MESSAGE” softkey.

Note that the CNC dynamically updates the active messages.

9.5 ACTIVE PAGES (SCREENS)

When selecting this option, the CNC will show the active page with the lowest number.

To delete a page or access the other active pages, the CNC will display the following softkey options:

NEXT PAGE	Press this softkey to display the next active page.
PREVIOUS PAGE	Press this softkey to display the previous active page.
CLEAR PAGE	Press this softkey to deactivate the page being displayed.

Note that the CNC dynamically updates the active pages.

9.6 SAVE PROGRAM

Press this softkey to save the PLC_PRG program into the user "Memkey Card" (CARD A).

The PLC program must be stopped before attempting to save it. If it is running, the CNC will ask whether it is desired to stop it or not.

The PLC program must be compiled, otherwise, the CNC will issue an warning message

If the PLC program is running, the CNC requests it to be stopped.

9.7 RESTORE PROGRAM

Press this softkey to restore (recover) the PLC program (PLC_PRG) from the user "Memkey Card" (CARD A) .

The PLC program must not be running any PLC program, otherwise, the CNC will ask whether it is desired to stop it or not.

After executing this instruction, the new source program recovered will replace the one that the PLC previously had. This new one must be compiled and started in order for the PLC to execute it.

9.8 RESOURCES IN USE

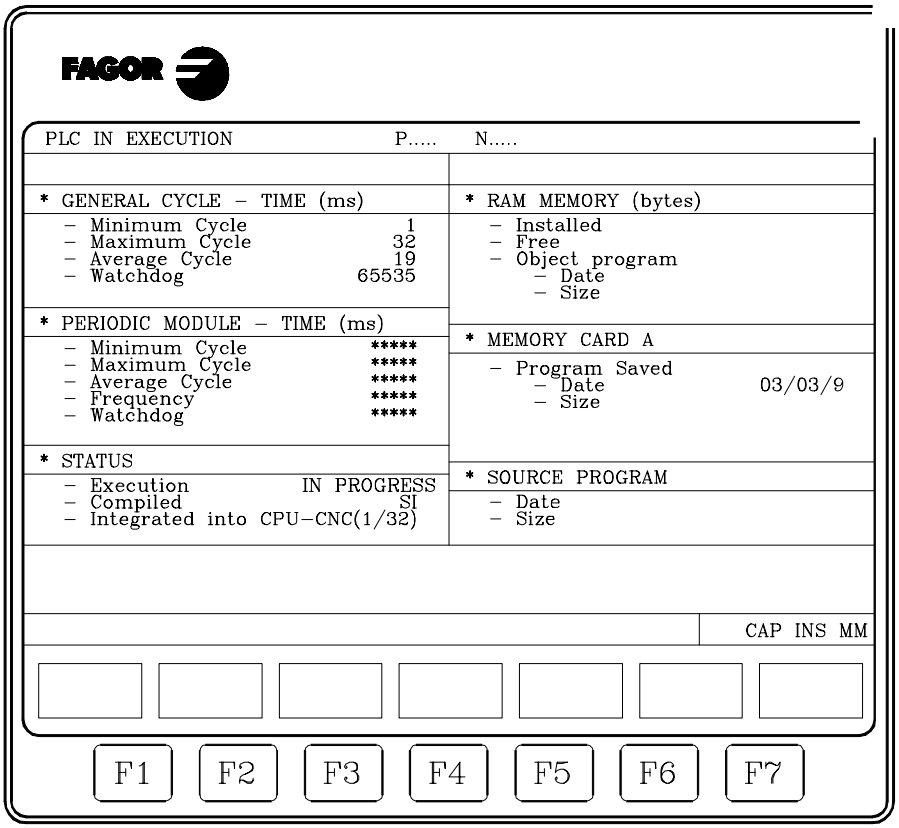
When selecting this option, the CNC will offer the softkeys to select the table of resources used in the PLC program.

The following resource tables are available:

INPUTS (I)
OUTPUTS (O)
MARKS (M)
REGISTERS (R)
TIMERS (T)
COUNTERS (C)

9.9 STATISTICS

This option shows the PLC memory distribution, the execution time of the various PLC modules, the PLC program status and the date when it was edited.



GENERAL CYCLE

This section shows the time (maximum, minimum and average) it takes the PLC to execute a program cycle.

This cycle includes:

* Updating the resources with the values of the physical inputs and internal CNC variables.

* Executing both the main cycle (PRG) and the periodic module.

* Updating the internal CNC variables and the physical outputs with the resource variables.

* Copying the resources into their corresponding images.

This section also shows the watchdog time selected by the PLC machine parameter "WDGPRG".

PERIODIC MODULE

This section shows the time (maximum, minimum and average) that it takes to execute the periodic module of the PLC.

It also shows the period assigned to this module by means of the directive instruction “PE t”. This period indicates how frequently the periodic module will be executed (every “t” milliseconds).

It also shows the watchdog time for this module selected by the PLC machine parameter “WDGPER”.

STATUS

Provides information on the PLC program status indicating whether it is compiled or not and whether it is stopped or in execution.

When the PLC does not have its own CPU (integrated into CPU-CNC), it will also indicate the time that the CNC’s CPU dedicates to the PLC. This value Will defined by the PLC machine parameter “CPUTIME”.

RAM MEMORY

This section indicates the system’s RAM memory available for the exclusive use of the PLC (installed) and it also indicates how much free memory there is.

The object program (executable) is obtained when compiling the source program and is the one executed by the PLC. This section shows the date when it was generated and the RAM memory space it occupies (size).

MEMORY CARD A

This section also shows the date the PLC program (PLC_PRG) was saved into the "Memkey Card" and its size.

SOURCE PROGRAM

This section indicates the date when it was last edited and its size.

The PLC source program is stored in the CNC’s RAM memory.

9.10 LOGIC ANALYZER

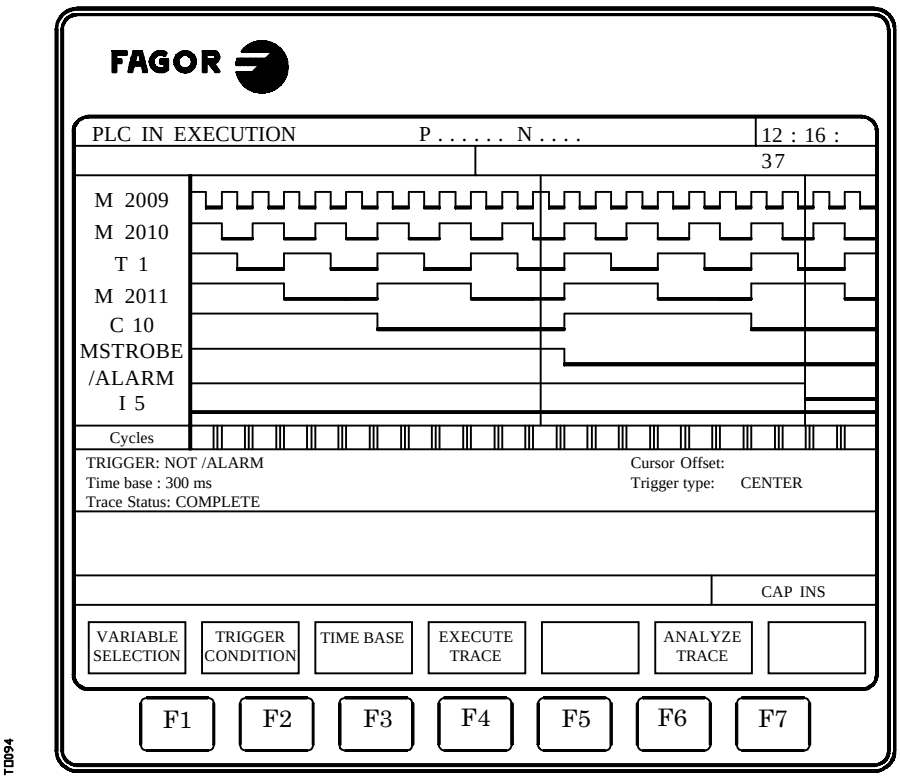
The logic analyzer is especially indicated to perform the machine setup and to determine errors and critical situations in the behavior of the various signals.

With this option it is possible to analyze the behavior of the logic signals of the PLC according to a time base and some trigger conditions established by the user.

Up to 8 signals can be monitored simultaneously. The results are displayed using a graphic interface to simplify the interpretation of the obtained data.

9.10.1 DESCRIPTION OF THE WORK SCREEN

The screen for the logic analyzer can be divided into the following display windows or areas:



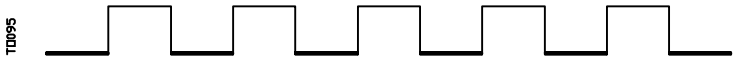
TD094

1.- Status window

It displays the graphic representation of the status of each one of the selected signals.

* The variable area shows the names or symbols of the logic signals to be analyzed.

* The status area shows the status of each variable in the shape of square waves. The line corresponding to logic level 0 is shown with a thicker line.



Also, a vertical red line is displayed to indicate the TRIGGER point and a vertical green line indicating the cursor position.

The green cursor line can be slid right and left along the trace and it can be used to measure the time difference between two of its points.

The status area is divided in several vertical sections. Each of them represents the amount of time established by the "time base" constant.

This constant determines the resolution of the logic signals and, after being defined by the user, can be modified at will. The relationship between the "time base" and the signal resolution is inversely proportional in such way that the smaller the time base, the greater the signal resolution is and vice versa.

2.- Cycle window

This window displays a series of vertical lines "|". Each one of them indicates the instant when a new PLC program cycle starts being executed.

It allows to maintain a relationship between the flow of the logic signals and the duration of each PLC execution cycle.

3.- Information window

This window provides general information about the trace being shown at the time. The shown data is the following:

Trigger — It shows the trigger condition set by the user to do the trace.

Time Base — Indicates the time base set by the user and used to show the current trace.

Trace Status — Indicates the current trace status. The shown texts and their meanings are as follows:

Empty	There is no calculated trace.
Capturing	There is one trace in progress.
Complete	One stored trace is available.

Cursor Offset Indicates the time difference, in milliseconds, between the cursor position (green line) and the trigger position (red line).

Trigger Type Indicates the type of trigger selected. The texts shown and their meanings are the following:

Before	The trigger is positioned at the beginning of the trace.
After	The trigger is positioned at the end of the trace.
Center	The trigger is positioned at the center of the trace.
Default	When no trigger condition has been specified.

4.- Editing window

It is the standard CNC editing window. It is used for all the processes requiring data entry.

5.- Message window

The CNC uses this window to display a warning or error message.

9.10.2 SELECTION OF VARIABLES AND TRIGGER CONDITIONS

Before requesting a trace, it is necessary to define the variables to be analyzed, the trigger type and conditions and the time base to be used to display the captured data.

To do this, the following softkey options are available: "**VARIABLE SELECTION**", "**TRIGGER CONDITION**" and "**TIME BASE**".

9.10.2.1 VARIABLE SELECTION

With this option it is possible to select up to 8 variables to be analyzed later.

It displays a cursor over the variable area and it can be slid up and down by means of the up and down arrow keys. The following softkey options will appear:

EDIT

With this option it is possible to edit a new variable or modify one of the currently defined variables.

Before pressing this softkey, we must select, with the cursor, the location where that variable will be shown.

Once this option is selected, the softkeys will change their background color to white and they will show the information corresponding to the editing type possible.

It is possible to analyze any logic signal of the PLC (I3, B1R120, TEN 3, CDW 4, DFU M200, etc.) and it can be referred to by its name or by its associated symbol.

It is also possible to analyze logic expressions, formed with one or more consultations which must follow the syntax and rules used to write the PLC equations.

M100 AND (NOT I15 OR I5) AND CPS C1 EQ 100

Although it might seem difficult to understand the processing of expressions and consultations at a logic analyzer, it should be borne in mind that it could prove very useful when it comes to finding out the status of a whole expression..

It is **not** possible to use more than 16 flank (edge) detecting instructions (DFU and DFD) among all the selected variable definitions and trigger conditions.

By pressing the **ESC** key, the variable being edited will be deleted. From this point on, that variable can be edited again.

Once the variable has been edited, press the **ENTER** key. The new variable will appear in the cursor position inside the variable area.

Only the first 8 characters of the selected variable or expression are shown even when it has more than 8.

The cursor will position at the next variable which will be shown in the editing window, thus being possible to continue editing new variables.

To quit this option, the editing area must be empty. If it is not empty, delete its contents by pressing **ESC** and then press **ESC** again.

DELETE

Use this option to delete a variable.

Before pressing this softkey, use the cursor to select the variable to be deleted.

To delete more variables, repeat these steps for each one of them.

CLEAR ALL

This option deletes all variables from the status window.

9.10.2.2 SELECTION OF TRIGGER CONDITION

The trigger condition as defined is that around which the data capture takes place. This data capture can be done before after or both before and after having met the selected trigger condition.

With this option it is possible to select the trigger type and condition of the logic analyzer. To do this, the following softkey options appear:

EDIT

With this option it is possible to edit the trigger condition around which the data capture will take place.

Once this option is selected, the softkeys will change their background color to white and they will show the information corresponding to the editing type possible.

It is possible to analyze logic expressions, formed with one or more consultations which must follow the syntax and rules used to write the PLC equations.

Examples of expressions and trigger conditions:

M100	The trigger occurs when M100 = 1
NOT M100	The trigger occurs when M100 = 0
CPS R100 EQ 1	The trigger occurs when R100 = 1
NOT I20 AND I5	The trigger occurs when the expression is true

It is **not** possible to use more than 16 flank (edge) detecting instructions (DFU and DFD) among all the selected variable definitions and trigger conditions.

By pressing the **ESC** key, the trigger condition being edited will be deleted. From this point on, that condition can be edited again.

Once the trigger condition has been edited, press ENTER. The new trigger condition will appear at the information window.

If no trigger condition has been specified, the system assumes one by default and it displays the message: "Trigger type: DEFAULT" in the information window. Besides, it will not permit the selection of any other possible types of trigger (before, center or after).

TRIGGER BEFORE

The CNC starts the data capture once (**after**) the selected trigger condition is met.

Then, once the trace has been executed, the trigger (vertical red line) will be positioned at the beginning of the trace.

TRIGGER AFTER

The CNC starts the data capture at the very instant the user selects the option to execute the trace **(before** the trigger condition is met).

The trace will be considered done when the selected trigger condition is met.

The trigger (vertical red line) will be positioned at the end of the trace.

TRIGGER CENTER

The CNC starts the data capture at the very instant the user selects the option to execute the trace.

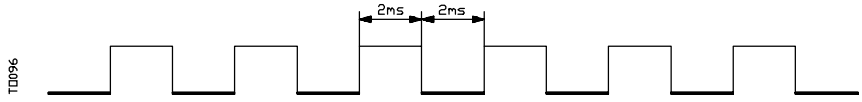
Then, once the trace has been executed, the trigger (vertical red line) will be positioned in the center of the trace.

9.10.2.3 SELECTION OF TIME BASE

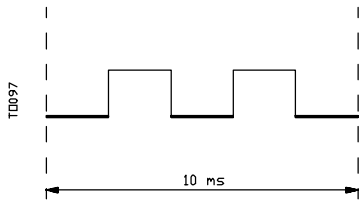
By means of this parameter, the user specifies the amount of time represented by each of vertical intervals.

Since the CRT width of these intervals is always the same, the signal resolution will be established by this time base in such way that the smaller the time base, the greater the signal resolution will be.

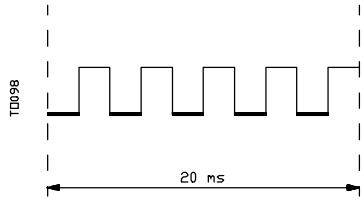
Example: Having a Mark whose status changes every 2 milliseconds.



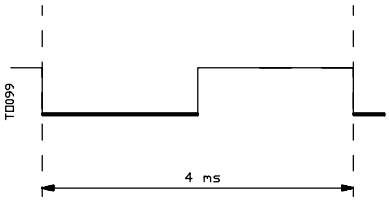
With a time base of 10 milliseconds, it will appear as follows:



With a time base of 20 milliseconds, it will appear as follows:



With a time base of 4 milliseconds, it will appear as follows:



The time base is given in milliseconds and the information window will show the selected value. By default, the CNC assumes a time base of 10 milliseconds.

It is possible to set a time base equal to the frequency of the signal to be monitored and then change it to obtain a finer signal resolution when analyzing the trace.

9.10.3 EXECUTE TRACE

Once having selected the variables and trigger conditions desired, press the "**EXECUTE TRACE**" softkey to indicate to the CNC to begin the data capture.

When the selected trigger condition is met, the trigger line displayed at the information window will change its color.

While the trace is being executed, the information window will display the message: "Trace Status: CAPTURING".

The trace will be completed when the internal memory buffer, dedicated to this function, is full or it is interrupted by pressing the "STOP TRACE" softkey. At this point, the information window will show the message: "Trace Status: COMPLETE".

9.10.3.1 DATA CAPTURE

The data capture takes place at the beginning of each cycle (PRG and PE), after reading the physical inputs and updating the marks corresponding to the CNC logic outputs and just before starting the PLC program execution.

Use this instruction to carry out another data capture while executing the PLC cycle.

This instruction permits the data capture of signals changing at frequencies greater than the cycle time as well as of those changing status during the execution of the cycle while keeping it the same at the beginning and at the end of the cycle.

Example of how to use the "**TRACE**" instruction:

```
PRG
   ----------
   ----------
   TRACE        ; Data capture
   ----------
   ----------
   TRACE        ; Data capture
   ----------
   ----------
   TRACE        ; Data capture
   ----------
   ----------
END

PE5
   ----------
   TRACE        ; Data capture
   ----------
END
```

The data capture in the execution of the trace in this program takes place:

- At the beginning of each **PRG** cycle
- Every time the periodic cycle (**PE**) is executed (every 5 milliseconds)
- 3 times while executing the **PRG** module.
- Once while executing the **PE** module.

This way, by means of the "**TRACE**" instruction the data capture can be done any time, especially at those program points considered more critical.

This instruction must only be used when debugging the PLC program and it should be avoided once the PLC program is fully debugged.

9.10.3.2 MODES OF OPERATION

The way the data is captured depends on the type of trigger selected. This section describes the different types of trigger being used as well as the way the data capture is done in each case.

Trigger Before

The data capture begins as soon as the selected trigger condition is met, that is when the trigger line shown at the information window changes its color.

The trace will be completed when the trace buffer is full or when the user interrupts it with the "**STOP TRACE**" softkey.

If interrupted before the trigger occurs, the trace will be empty.

Trigger after

The data capture begins the instant the user presses the "**EXECUTE TRACE**" softkey.

The trace will be completed when the selected trigger condition is met or it is interrupted by pressing the "**STOP TRACE**" softkey.

If interrupted before the trigger occurs, a trace will be shown with data but without the trigger position (vertical red line).

Trigger center

The data capture begins the instant the user presses the "**EXECUTE TRACE**" softkey.

The CNC will enable half the trace buffer to store the data corresponding to the trace prior to the trigger and the other half for the data corresponding to the trace after the trigger.

The trace is completed when its buffer is full or when it is interrupted by pressing the "**STOP TRACE**" softkey.

If interrupted before the trigger occurs, a trace will be shown with data but without the trigger position (vertical red line).

Trigger by Default

The CNC carries out this type of trace when no trigger condition has been specified.

The data capture begins the instant the "**EXECUTE TRACE**" softkey is pressed.

The trace is completed when interrupted by pressing the "**STOP TRACE**" showing a trace with data but without the trigger position (vertical red line).

9.10.3.3 TRACE REPRESENTATION

Once the data capture is done, the CNC will display graphically in the status window the status of the signals based on the trace calculated for the analyzed variables.

Also, a vertical red line indicating the trigger position and a vertical green line indicating the cursor position will appear superimposed on the trace.

The cursor position (vertical green line) can be slid along the trace by means of the following keys:

Left arrow Moves the cursor one pixel to the left.

While keeping this key pressed, the cursor will advance automatically one pixel at a time and increasing its speed.

If the cursor is positioned at the left end, the trace will be shifted to the right while the cursor stays in the same position.

Right arrow Moves the cursor one pixel to the right.

While keeping this key pressed, the cursor will advance automatically one pixel at a time and increasing its speed.

If the cursor is positioned at the right end, the trace will be shifted to the left while the cursor stays in the same position.

Previous page Moves the cursor one screen to the left.

Next page Moves the cursor one screen to the right.

The CNC will show at all times, in the information window, the cursor position (vertical green line) with respect to the trigger position (vertical red line). This information will appear as “Cursor Offset” and it will be given in milliseconds.

9.10.4 ANALYZE TRACE

Once the data capture is done, the CNC, besides displaying the status window, will enable the "**ANALYZE TRACE**" softkey.

With this option it is possible to position the cursor (vertical green line) at the beginning of the trace, at the end of it or at a specific point along the trace. It is also possible to change the time base for the trace or calculate the time difference between two points of the trace.

To do this, the following softkey options are available:

Find beginning The cursor will position at the beginning of the trace being shown.

Find End It will show the last section of the trace and the cursor will position at the end of it.

Find Trigger It will show the area of the trace corresponding to the trigger zone. The trigger position will appear as a vertical red line over the trace.

The CNC will execute this option when a trigger occurs while analyzing the trace.

Find Time Base When pressing this key, the CNC will request the cursor position with respect to the trigger point. This value is given in milliseconds.

For example: Having selected a "Find time base" of -1000 milliseconds, the CNC will show the trace section corresponding to 1 second prior to the trigger instant.

If no trigger occurred while analyzing the trace, the CNC will assume that the indicated position is referred to the beginning of the trace.

Calculate Times With this option it is possible to find out the time between two points of the trace. To do this, follow these steps in order to set the initial and final points of the calculation.

Position the cursor at the initial point of calculation and press the "**MARK BEGINNING**" softkey to validate it. Use the "left arrow", "right arrow", "page-up" and "page down" keys to move the cursor.

Position the cursor at the final point of calculation and press the "**MARK END**" softkey to validate it.

The CNC will display in the message window the time difference between those two points. It will be given in milliseconds.

This feature can prove very useful to calculate exactly the rise and fall times of a signal, times between two signals, times between the trigger of a signal and the beginning of a cycle, etc.

Modify Time Base This option permits the "Time Base" to be modified.

The status area is divided into several vertical sections. Each of these sections represents a time pitch determined by the "Time Base" constant.

The relationship between the "Time Base" and the signal resolution is inversely proportional in such way that the smaller the "time base", the greater the signal resolution and vice versa.

When pressing this softkey, the CNC will request the new value for the time base. This value must be given in milliseconds.

10. SCREEN EDITOR

In this operating mode, the operator can create up to 256 pages (screens) which will be stored in the "Memkey Card".

The operator can also create up to 256 SYMBOLS to be used when creating the user screens. These symbols are also stored in the "Memkey Card".

The information contained in a page or symbol cannot occupy more than 4Kb of memory. Otherwise, the CNC will issue the corresponding error message.

The user screens stored in the "Memkey Card" may be:

* Used in the screen customizing programs as described next.

* Displayed on power-up (page 0) instead of the FAGOR logo.

* Activated from the PLC.

  The PLC has 256 marks, with their corresponding mnemonics, to select the user screens. These marks are:

M4700	PIC0
M4701	PIC1
M4702	PIC2
———	———
———	———-
M4953	PIC253
M4954	PIC254
M4955	PIC255

  When any of these marks is set high, its corresponding screen (page) is activated.

* Used to complete the M function assistance system (screens 250-255).

  When requesting programming assistance for the auxiliary M functions by pressing the [HELP] key, the CNC will show the corresponding internal screen (page).

  When user page 250 is defined, that information will also include the symbol indicating that more help pages are available. By pressing this key, the CNC will display user screen 250.

  The CNC will keep showing that indicator as long as there are more user screens defined (250-255).

  These screens must be defined in a row always starting from page 250. If one of them is missing, the CNC will interpret that there are no more screens defined.

The user screens activated from the PLC may be displayed with the ACTIVE PAGES option of the PLC.

The various options available in this operating mode are:

* **UTILITIES** to manipulate user symbols and screens (edit, copy, delete, etc.).

* **GRAPHIC ELEMENTS** to insert graphic elements in the selected symbol or screen.

* **TEXTS** to insert texts in the selected symbol or screen.

* **MODIFICATIONS** to modify the selected symbol or screen.

10.1 UTILITIES

The various options available in this mode are:

DIRECTORY

To display the directory of user screens and symbols that are stored in the "Memkey Card" (CARD A) or in external devices through the serial lines.

Select the desired device and directory.

The CNC shows the size (in bytes) of each user screen (page) and symbol.

COPY

To make copies within the "Memkey Card" (CARD A) or between the "CARD A" and the external devices.

Examples:

to copy screen (page) 5 from the "Memkey Card" to serial line 2

COPY PAGE 5 IN SERIAL LINE 2 (DNC)

to copy screen (page) 50 from serial line 2 into the "Memkey Card"

COPY SERIAL LINE 2 (DNC) IN PAGE 50 ENTER

to copy symbol 15 as symbol 16 within the "Memkey Card"

COPY SYMBOL 15 IN SYMBOL 16 ENTER

DELETE

To delete a screen or symbol from the "Memkey Card". To do that, proceed as follows:

- Press the [DELETE] softkey
- Press the [PAGE] or [SYMBOL] softkey
- Key in the page or screen number to be deleted and press [ENTER]

The CNC will request confirmation of the command.

RENAME

To assign a new name or comment to a page or symbol of the "Memkey Card".

If there is another one with the same number, the CNC will display a warning message and it will offer the chance to modify the command.

Examples:

to change the page number from 20 to 55

RENAME PAGE 20 TO NEW NUMBER 55 ENTER

to change the comment of symbol 10

RENAME SYMBOL 10 TO NEW COMMENT "Test" ENTER

EDIT

To edit a new user screen (page) or symbol proceed as follows:

- Press the [EDIT] softkey
- Press the [PAGE] or [SYMBOL] softkey
- Key in the page or symbol number
- Press [ENTER]

If the page or symbol does not exists, an empty page will appear in the editing area.

How to edit user screens and symbols is described later on in this chapter.

If the selected screen or symbol has been changed, the CNC will request whether it is to be saved or not in the following instances:

- When exiting the screen editor.
- When selecting another screen (page) or symbol.

SAVE

To save the page or symbol being edited into the "Memkey Card".

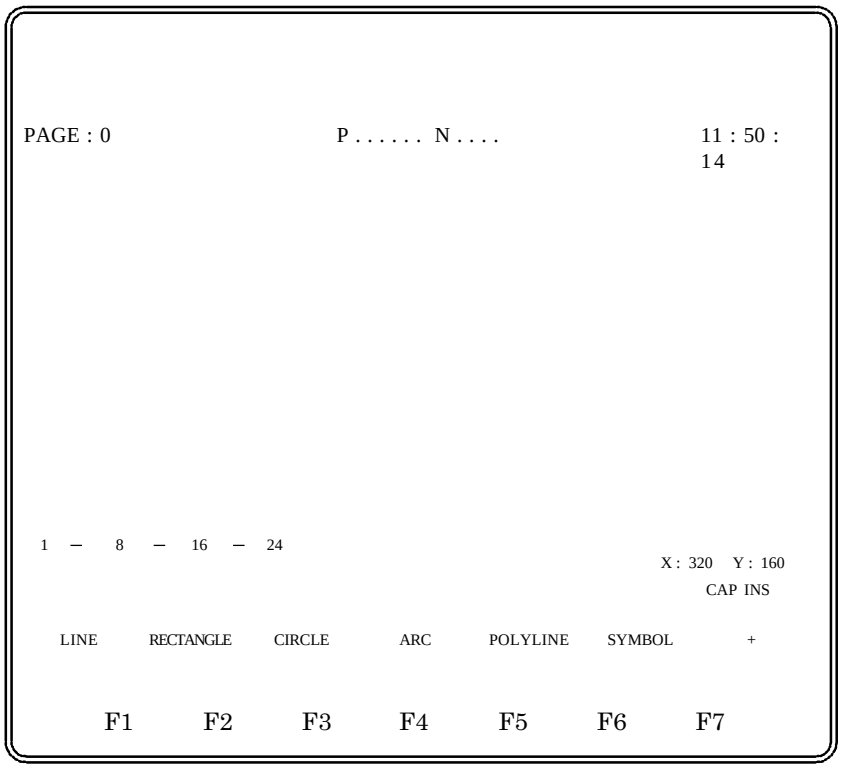
10.2 EDITING CUSTOM SCREENS (PAGES) AND SYMBOLS

In order to edit a page or symbol, it is necessary to selected first by means of the EDIT option of the UTILITIES mode of operation.

To edit or modify a page or symbol, use the options: **GRAPHIC ELEMENTS, TEXTS**, and **MODIFICATIONS**.

The information contained in a page or symbol must not occupy more than 4Kb; otherwise, the CNC will issue the corresponding error message.

Once the page or symbol has been selected, the CNC will display a screen similar to this



one:

* The upper left-hand side of the screen will show the number of the page or symbol being edited.

* The main window will show the selected page or symbol. When it is a new page or symbol, the main window will be “blank” (blue background).

* There is also a window at the bottom of the screen which shows the different editing parameters and highlights their selected values.

The various parameters available are:

* The type of drawing line used when defining the graphic elements.

* The cursor moving steps (cursor advance) in pixels.

* The letter size to create the texts for the pages and symbols.

* The background and foreground (main) colors for the graphic elements and for the letters.

  One of the color rectangles shown has another rectangle in it. The inside rectangle indicates the selected main color and the outside rectangle indicates the selected background color.

This window also shows The cursor position coordinates in pixels. The horizontal position is indicated by the X value (1 through 638) and the vertical position by the Y value (0 through 334).

Once one of the options (**GRAPHIC ELEMENTS, TEXTS** or **MODIFICATIONS**) has been selected, it will be possible to modify the editing parameters any time.

This way, it will be possible to edit texts and shapes of different color and size.

Press **INS** to access this menu.

Once in this mode, the CNC will show the softkeys corresponding to the various options to modify these parameters. These options are described next.

Press **INS** again to quit this mode and return to the previous menu.

CURSOR ADVANCE

With this option it is possible to select the cursor moving step in pixels (1, 8, 16, 24).

Follow these steps after pressing this softkey:

1.- Use the right and left arrow keys to select the desired step.

The currently selected step will be highlighted.

2.- Press **ENTER** to validate the selected step or **ESC** to quit this mode leaving the previous selection intact.

When editing a new page or symbol, the CNC assumes the default value of 8.

TYPE OF LINE

With this option it is possible to select the type of line used to define the graphic elements.

Follow these steps after pressing this softkey:

1.- Use the right and left arrow keys to select the desired type of line.

The currently selected line type will be highlighted.

2.- Press **ENTER** to validate the selected step or **ESC** to quit this mode leaving the previous selection intact.

When editing a new page or symbol, the CNC assumes the “fine line” by default.

It is not possible to use the thick line to draw polylines or polygons. They are always drawn in fine line.

TEXT SIZE

With this option it is possible to select the size of the letters used to write the texts to be inserted in the pages or symbols.

Three sizes are available:

* Normal size.

  All the characters of the keyboard, numbers, signs, upper and lower case letters, can be written in this size.

* Double and triple sizes.

  Only capital letters **A through Z**, numbers **0 through 9** ; the “*****”, “**+**”, “**-**”, “**.**”, “**:**” , "**#**", "**%**", "**/**", "**<**", "**>**", "**?**" signs and the special characters: "**Ç**", "**Ä**", "**Ö**", "**Ü**", "**ß**" can be written in these sizes.

  When selecting lower case letters for these sizes, the CNC will convert them automatically into upper case.

Follow these steps to select the text size after pressing this softkey:

1.- Use the right and left arrow keys to select the desired size.

The currently selected size will be highlighted.

2.- Press **ENTER** to validate the selected step or **ESC** to quit this mode leaving the previous selection intact.

When editing a new page or symbol, the CNC assumes the normal size by default.

BACKGROUND COLOR

With this option it is possible to select the background color over which the different graphic elements and texts will be edited.

It is not possible to select the background color when editing a symbol since it is an attribute of the page and not of the symbol. Therefore, when inserting a symbol into a page, the symbol will take the background of that page.

If the desired background color is **WHITE**, it is recommended to use a different color while creating the page since the cursor the "drawing" cursor is always white and will become invisible with this background color. Once the complete page (screen) is created, the background color can be changed to the desired one.

One of the color rectangles shown has another rectangle in it. The inside rectangle indicates the selected main color and the outside rectangle indicates the selected background color.

To select the background color, follow these steps:

1.- Use the right and left arrow keys to select the desired color among the 16 shown.

The CNC will show the background color being selected by placing the main-color rectangle inside the rectangle corresponding to the background color being selected.

2.- Press **ENTER** to validate the selected color or **ESC** to quit this mode leaving the previous selection intact.

When editing a new page or symbol, the CNC assumes a blue background color by default.

MAIN COLOR

With this option it is possible to select the color used to draw and write texts on the page (screen) or symbol.

One of the color rectangles shown has another rectangle in it. The inside rectangle indicates the selected main color and the outside rectangle indicates the selected background color.

To select the main color, follow these steps:

1.- Use the right and left arrow keys to select the desired color among the 16 shown.

The CNC will show the main color being selected by placing a white inside rectangle. It will also display the rectangle containing both the selected background color and the main color being selected here.

2.- Press **ENTER** to validate the selected color or **ESC** to quit this mode leaving the previous selection intact.

When editing a new page or symbol, the CNC assumes white as the main color by default.

GRID

This softkey superimposes a grid over the screen in order to facilitate the lay out of the different components of the page or symbol being created or modified. This grid is formed by white or black points (depending on the background color) separated 16 pixels from each-other.

The grid points will be white when the selected background color corresponds to one of the 8 upper color rectangles and they will be black when the selected background color corresponds to one of the 8 lower color rectangles.

Press this softkey again to get rid of the grid.

Every time the grid is displayed, the CNC will reset the cursor advance (step) to 16 pixels.

Therefore, the cursor will move from grid point to grid point every time the arrow keys are pressed to position it on the screen. However, the cursor advance may be modified afterwards by selecting it with the **CURSOR ADVANCE** softkey.

10.3 GRAPHIC ELEMENTS

Before accessing this option, it is necessary to select the page or symbol to be edited or modified by means of the EDIT option of the UTILITIES mode of operation.

With this option it is possible to include graphic elements in the selected page or symbol. The CNC displays a screen 80 columns wide (640 pixels for X coordinate) by 21 rows high (336 pixels for Y coordinate).

When editing a new page, the CNC will position the cursor in the center of the screen and when editing a new symbol, it will position it at the upper left-hand corner.

The cursor is white and can be moved around with the up and down arrow keys and the left and right arrow keys.

The cursor can also be moved by using the following keystroke combinations:

SHIFT → Positions the cursor at the last column (X638)

SHIFT ← Positions the cursor at the first column (X1)

SHIFT ↑ Positions the cursor at the first row (Y0).

SHIFT ↓ Positions the cursor at the last row (Y334).

It is also possible to key in the XY coordinates of the point where the cursor is to be positioned. To do this, follow these steps:

* Press “X” or “Y”.

  The CNC will highlight, in the editing parameter display window, the cursor position along the selected axis (column or row).

* Key in the position value corresponding to the point where the cursor is to be placed along this axis.

  The horizontal position is defined as the X value between 1 and 638 and the vertical position as the Y value between 0 and 334.

  Once these coordinates have been keyed in, press **ENTER** and the CNC will position the cursor at the indicated coordinates.

Once this option is selected, it is possible to modify the editing parameters at any time even while defining the graphic elements. This way, it is possible to edit shapes of different line and color.

Press **INS** to access this menu.

Once in this mode, press the corresponding softkey to modify those parameters.

Press **INS** again to quit this mode and return to the previous menu.

The possible graphic elements which can be used to create a page or symbol are selected with the softkeys and are the following:

LINE

Follow these steps after pressing this softkey:

1.- Place the cursor at the beginning of the line and press **ENTER** to validate it.

2.- Move the cursor to the end of the line (the CNC will continuously show the line being drawn).

3.- Press **ENTER** to validate the line or **ESC** to cancel it.

Repeat the preceding steps to draw more lines. If no more lines are desired, press **ESC** to return to the previous menu.

RECTANGLE

Follow these steps after pressing this softkey:

1.- Place the cursor on one of the corners of the rectangle and press **ENTER** to validate it.

2.- Move the cursor to the opposite corner. The CNC will continuously show the rectangle being drawn.

3.- Press **ENTER** to validate the rectangle or **ESC** to cancel it.

Repeat these steps to draw more rectangles. If no more rectangles are desired, press ESC to return to the previous menu.

CIRCLE

Follow these steps after pressing this softkey:

1.- Place the cursor at the center of the circle and press **ENTER** to validate it.

2.- Move the cursor in order to define the radius. As the cursor moves, the CNC will show the circle corresponding to that radius.

3.- Press **ENTER** to validate the circle or **ESC** to cancel it.

Once the circle is validated, the cursor is positioned at its center in order to facilitate the drawing of concentric circles.

Repeat these steps to draw more circles. If no more circles are desired, press **ESC** to return to the previous menu.

ARC

Follow these steps after pressing this softkey:

1.- Place the cursor at one of the arc's ends and press **ENTER** to validate it.

2.- Move the cursor to the other end of the arc (the CNC will show a line joining both ends) and press **ENTER** to validate it.

The cursor is now positioned automatically at the center of that line.

3.- Move the cursor to define the curvature. The line will become an arc passing through 3 points (the two ends and the cursor point).

4.- Press **ENTER** to validate it or **ESC** to cancel it.

Repeat these steps to draw more arcs. If no more arcs are desired, press ESC to return to the previous menu.

POLYLINE

A polyline consists of several lines where the last point of one of them is the beginning point for the next one.

Follow these steps after pressing this softkey:

1.- Place the cursor at one of the ends of the polyline and press **ENTER** to validate it.

2.- Move the cursor to the end of the first line (which will be the beginning of the next one). The CNC will continuously show the line being drawn.

Press **ENTER** to validate the line or **ESC** to quit this option (which will delete the complete polyline).

3.- Repeat steps 1 and 2 for the rest of the lines.

Note that the maximum number of lines in a polyline is 127.

Once the polyline is drawn, press **ENTER** again to validate it or **ESC** to quit this option deleting the complete polyline.

Repeat these steps to draw more polylines and if no more polylines are desired, press ESC to return to the previous menu.

SYMBOL

This option allows a symbol to be drawn in the page or symbol being edited.

After pressing this softkey, the following steps will be taken.

1.- Enter the number of the symbol to include in the page or symbol being edited and press the ENTER key to validate it.

The CNC will show the cursor situated at the reference point corresponding to the symbol (upper left hand corner of the symbol).

2.- Move the cursor to the position where it is required to place the symbol. In this move, only the cursor will move and not the symbol.

3.- Press the ENTER key to validate it or the ESC key if you wish to quit.

Once the symbol has been validated the CNC will show it in the place indicated.

4.- To include more symbols, repeat the above operations.

5.- Press the ESC key to quit and go back to the previous menu.

If a symbol is being edited this symbol cannot be included in itself. Therefore, if symbol 4 is being edited, any symbol can be included except symbol 4.

Warning:

If a symbol is deleted, the CNC will update all the pages or symbols that contain it because all the calls to it will remain active.

When displaying a page or symbol which has a call to a nonexistent symbol (deleted or not defined), that area of the page will appear blank.

If this symbol is edited again later, the new representation assigned to the symbol will appear in all the pages and symbols which contain a call to it.

POLYGON

A polygon is a closed polyline whose beginning and end points coincide.

After pressing the softkey, the following steps will be taken:

1.- Place the cursor on one of the vertices of the polygon and press the ENTER key to validate it.

2.- Move the cursor to the following vertex of the polygon (the CNC will show the line you are trying to draw).

Press the ENTER key to validate the line or the ESC key if you wish to abandon.

3.- Repeat step 2 for the remaining vertices.

Once all vertices are defined, press the ENTER key and the CNC will complete the polygon or the ESC key if you wish to quit.

The maximum number of sides on the polygon is limited to 127.

FILLED POLYGON

After pressing this softkey, follow the steps as in the POLYGON option, but in this case, after completing the definition of the polygon it will be filled with the color used for its definition.

FILLED CIRCLE

After pressing this softkey follow the steps as in the CIRCLE option, but in this case, after completing the definition of the circle it will be filled with the color used for its definition.

FILLED RECTANGLE

After pressing this softkey follow the steps as in the RECTANGLE option, but in this case, after completing the definition of the rectangle it will be filled with the color used for its definition.

10.4 TEXTS

Before accessing this option, it is necessary to select the page or symbol to be edited or modified by means of the EDIT option of the UTILITIES mode of operation.

With this option it is possible to include texts in the selected page or symbol. The CNC displays a screen 80 columns wide (640 pixels for X coordinate) by 21 rows high (336 pixels for Y coordinate).

When editing a new page, the CNC will position the cursor in the center of the screen and when editing a new symbol, it will position it at the upper left-hand corner.

The cursor is white and can be moved around with the up and down arrow keys and the left and right arrow keys.

The cursor can also be moved by using the following keystroke combinations:

SHIFT [→] Positions the cursor at the last column (X638)

SHIFT [←] Positions the cursor at the first column (X1)

SHIFT [↑] Positions the cursor at the first row (Y0).

SHIFT [↓] Positions The cursor at the last row (Y334).

It is also possible to key in the XY coordinates of the point where the cursor is to be positioned. To do this, follow these steps:

* Press “X” or “Y”.

  The CNC will highlight, in the editing parameter display window, the cursor position along the selected axis (column or row).

* Key in the position value corresponding to the point where the cursor is to be placed along this axis.

  The horizontal position is defined as the X value between 1 and 638 and the vertical position as the Y value between 0 and 334.

  Once these coordinates have been keyed in, press **ENTER** and the CNC will position the cursor at the indicated coordinates.

Once this option is selected, it is possible to modify the editing parameters at any time even while defining the graphic elements. This way, it is possible to edit texts of different size and color.

Press **INS** to access this menu.

Once in this mode, press the corresponding softkey to modify those parameters.

Press **INS** again to quit this mode and return to the previous menu.

It is also possible to insert one of the texts available at the CNC or a text previously keyed in by the user. To do this, the following softkey options are available:

USER DEFINED TEXT

Follow these steps to insert the desired text:

1.- Press **ENTER**.

The CNC will display a text editing window. The cursor within this window can be moved with right and left arrow keys.

2.- "Type" the desired text.

A rectangle will be displayed which will enlarge as the text is "typed" in the editing window thus indicating the screen space that this text will occupy.

Press **ESC** to cancel this option and the previous menu will be displayed.

3.- Press **ENTER** once the text has been correctly "typed in".

The typed text will remain in the editing window and the cursor will be positioned in the main window.

4.- Position the rectangle by moving the cursor.

5.- Press **ENTER** to validate this command and the text will replace the rectangle on the screen.

Note that once the text has been "entered", neither its size nor its color can be modified. Therefore, these options must be selected before pressing **ENTER**.

TEXT NUMBER

With this option it is possible to select a text used by the CNC itself in its various operating modes and insert it into the current page or symbol.

To insert one of these predetermined texts, follow these steps:

1.- Press the corresponding softkey.

The CNC will show a screen area to indicate the text number. The cursor may be moved within this area with the right and left arrow keys.

2.- Indicate the desired number by keying it in from the keyboard and press **ENTER**.

The CNC will display the text corresponding to this number and the rectangle indicating the screen space it occupies.

If another text is desired, key in the other number and press **ENTER** again.

Press **ESC** to quit this option without inserting the text and the CNC will show the previous menu.

3.- Once the desired text has been selected, press **ENTER**.

The typed text will remain in the editing window and the cursor will be positioned in the main window.

4.- Position the rectangle by moving the cursor.

5.- Press **ENTER** to validate this command and the text will replace the rectangle on the screen.

Observe that once the text has been "entered", neither its size nor its color can be modified. Therefore, these options must be selected before pressing **ENTER**.

Warning:

This application may be useful when the pages or symbols being edited are to be shown in other languages since the CNC will translate them into the chosen language.

Usually, when the texts are to be shown in one single language, it is more practical to simply write them up instead of searching them in a list of more than 1500 predetermined messages.

However, should anyone desire the printout of these predetermined texts, feel free to request it from Fagor Automation.

10.5 MODIFICATIONS

Before accessing this option, it is necessary to select the page or symbol to be edited or modified by means of the EDIT option of the UTILITIES mode of operation.

With this option it is possible to include texts in the selected page or symbol. The CNC displays a screen 80 columns wide (640 pixels for X coordinate) by 21 rows high (336 pixels for Y coordinate).

When editing a new page, the CNC will position the cursor in the center of the screen and when editing a new symbol, it will position it at the upper left-hand corner.

The cursor is white and can be moved around with the up and down arrow keys and the left and right arrow keys.

The cursor can also be moved by using the following keystroke combinations:

SHIFT Positions the cursor at the last column (X638)

SHIFT Positions the cursor at the first column (X1)

SHIFT Positions the cursor at the first row (Y0).

SHIFT Positions the cursor at the last row (Y334).

It is also possible to key in the XY coordinates of the point where the cursor is to be positioned. To do this, follow these steps:

* Press “X” or “Y”.

  The CNC will highlight, in the editing parameter display window, the cursor position along the selected axis (column or row).

* Key in the position value corresponding to the point where the cursor is to be placed along this axis.

  The horizontal position is defined as the X value between 1 and 638 and the vertical position as the Y value between 0 and 334.

  Once these coordinates have been keyed in, press **ENTER** and the cursor will be positioned at the indicated coordinates.

The possible options to modify a page or symbol are:

CLEAR PAGE

Allows the selected page or symbol to be deleted.

Once this softkey has been pressed, the CNC will request an OK before executing the indicated operation.

If this option is executed, the CNC will delete the page or symbol being edited, but it will keep in the "Memkey Card" the contents of that page or symbol the last time the "SAVE” command was executed.

DELETE ELEMENTS

This option allows an element of the displayed page or symbol to be selected and then deleted.

To do this follow these steps:

1.- Place the cursor in the position to delete an element and press the **ENTER** key to validate it.

An area of between ± 8 pixels from the position indicated will be analyzed.

If the element to be deleted is a filled circle or a filled polygon, the cursor must be positioned on a point on the circumference or external polygon (periphery).

2.- If any graphic element or text exists in this area, this will be highlighted and you will be asked if you wish to delete it.

Press the ENTER key to delete this element, otherwise the ESC key.

Should there be several elements in this area, the CNC will highlight them in succession and it will ask for confirmation before deleting any of them.

MOVE SCREEN

With this option it is possible to reposition the whole page (not its individual elements separately) and it can only be used to move pages and not symbols.

It allows the entire page to be moved with the right, left, up and down arrow keys.

The center of the page is taken as a reference for this movement.

To do this follow these steps:

1.- The CNC will show the page with the cursor placed in the middle of the screen.

2.- Move the cursor to the position to place the page reference point.

Press ESC to quit this option without making any changes and the CNC will show the previous menu.

Repeat these steps to perform more moves, otherwise, press ESC and the CNC will show the previous menu.

11. MACHINE PARAMETERS

In order for the machine tool to execute the programmed instructions correctly, the CNC must know specific data on the machine such as feedrates, accelerations, feedbacks, automatic tool changes, etc.

This data is determined by the manufacturer of the machine and must be stored in the machine parameter tables.

These tables may be edited in this work mode or copied into the "Memkey Card" or a PC as described later on.

The CNC has the following groups of machine parameters:

* General machine parameters
* Axis parameters (one table per axis)
* Spindle parameters
* RS-422 and RS-232-C serial port configurations
* Ethernet configuration parameters
* PLC parameters
* M miscellaneous functions
* Leadscrew error compensation (one table per axis)
* Cross Compensations between two axes (for example: Beam sag).

First, the general machine parameters must be set as by means of these the machine axes are defined and therefore the Axis Parameter tables.

It must also be defined whether the machine has cross compensation and between which axes, and the CNC will generate the corresponding cross compensation parameters.

By means of the general machine parameters, the table lengths for the Tool Magazine, Tools, Tool Offsets and the miscellaneous M functions are defined.

By means of the Axis Parameters it is defined whether the axis has Leadscrew error Compensation or not and the length of the corresponding table.

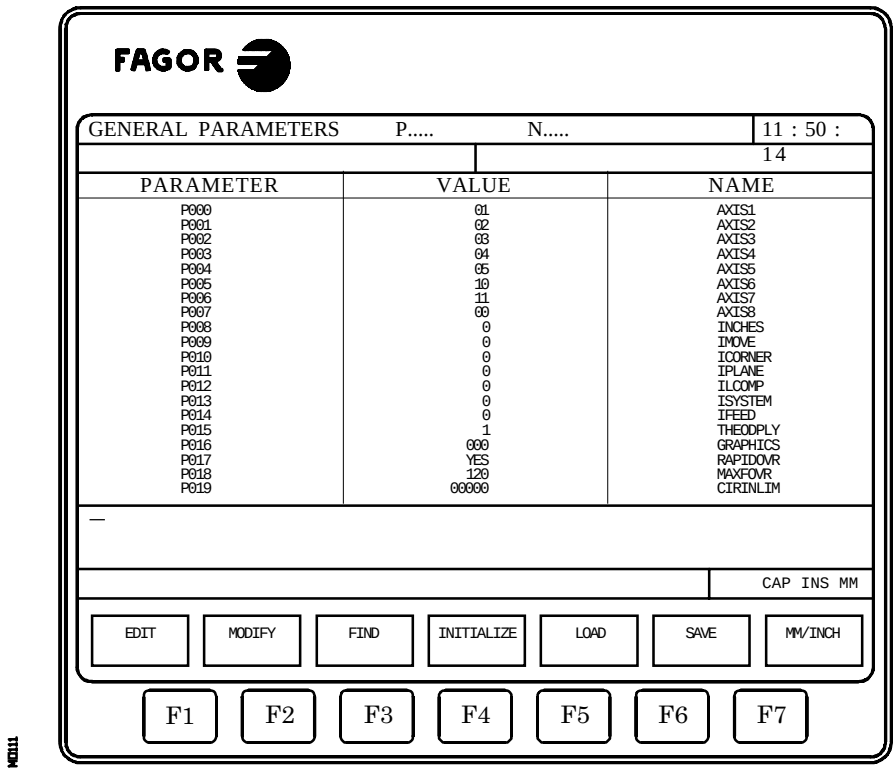
Once the general machine parameters are defined, press **SHIFT RESET** for the CNC to enable the required tables.

It is recommended to save the tables in the "Memkey Card" or out to a peripheral device or PC.

When accessing this operating mode, the CNC will show the tables that are saved in the "Memkey Card" (CARD A).

11.1 MACHINE PARAMETER TABLES

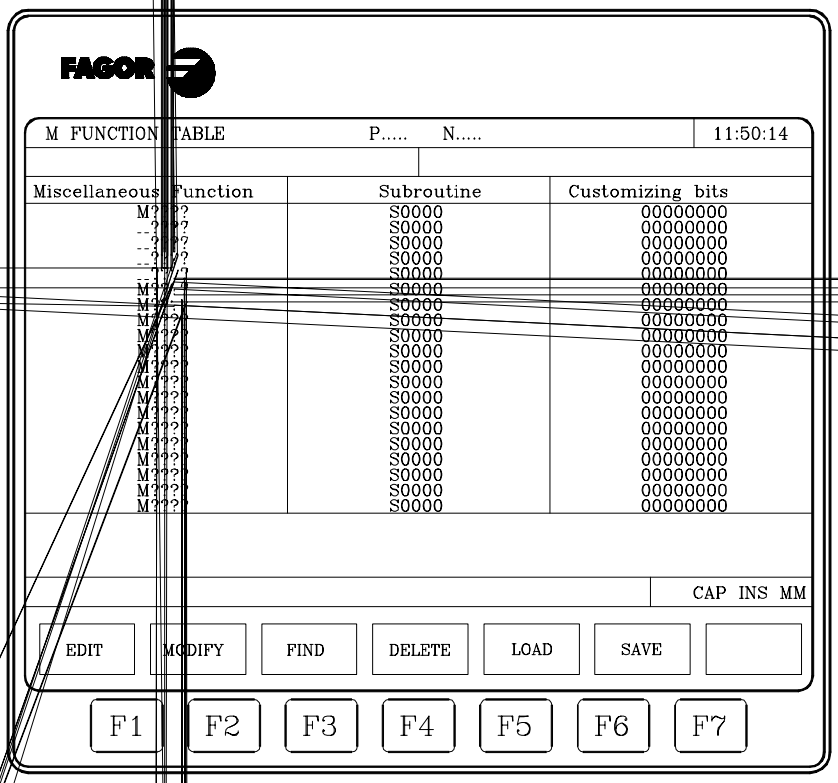
The General, Axis, Spindle, Serial ports and PLC tables have the following structure:



Where the parameter number is indicated, the value assigned to it and the name or mnemonic associated with this parameter.

11.2 MISCELLANEOUS FUNCTION TABLES

The table corresponding to the miscellaneous M functions has the following structure:



The number of M functions in the table is defined by means of the general machine parameter "NMISCFUN". The following is defined for each line:

* The number (0-9999) of the defined miscellaneous M functions:

  If an M function is not defined, the CNC will show M????.

* The number of the subroutine to be associated with this miscellaneous function.

* 8 customizing bits

x	x	x	x	x	x	x	x
7	6	5	4	3	2	1	0

  Bit 0 Indicates whether the CNC must (=0) or must not (=1) wait for the signal AUXEND (signal of the M executed) to resume program execution.

  Bit 1 Indicates whether the M function is executed before (=0) or after (=1) the movement of the block in which it is programmed.

  Bit 2 Indicates whether the execution of the M function interrupts (=1) or not (=0) the preparation of the blocks.

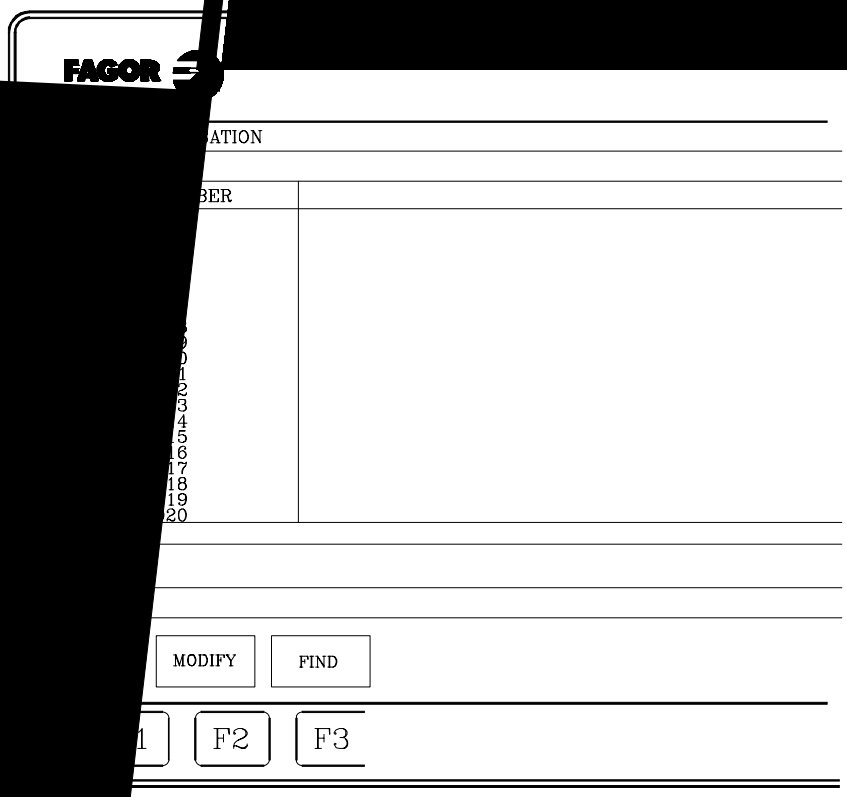
  Bit 3 Indicates whether the M function is executed after calling the associated subroutine (=0) or only the associated subroutine is executed (=1).

  Bit 4 When bit 2 is set to "1", it indicates whether block preparation is to be interrupted until the "M" function starts executing (=0) or until its execution is finished (=1).

  The rest of the bits are not being used at this time.

11.3 LEADSCREW

The tables for leadscr



f points of each of these is defined by means of the axis machine parameter The following is defined for each of line:

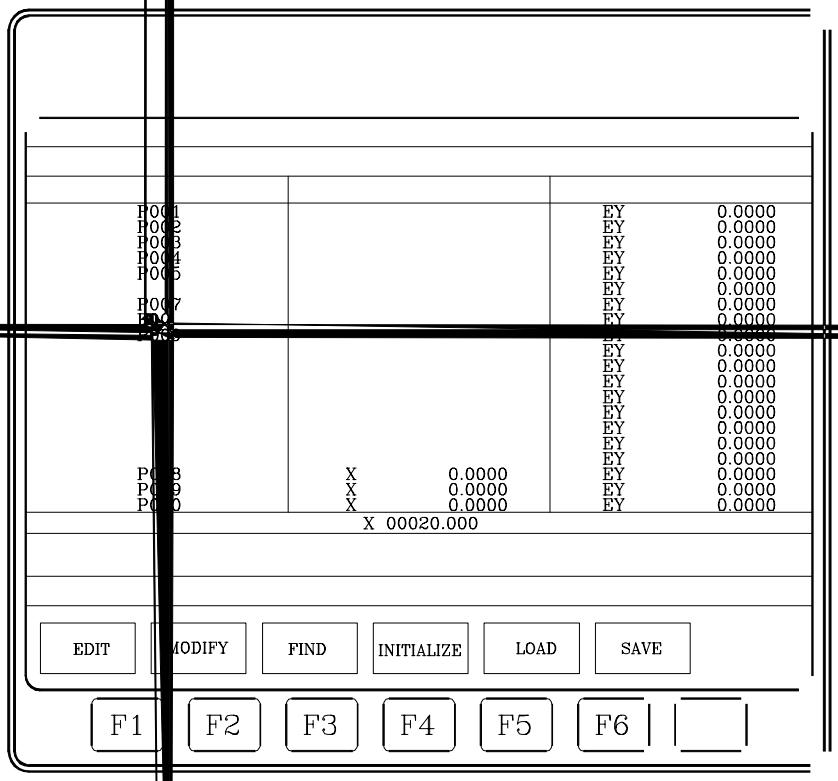
f the axis to be compensated.

his axis in this position.

rrent position of the selected axis is displayed and updated as the machine axis

11.4 CROSS COMPENSATION TABLES

The tables corresponding to cross compensation have the following structure.



The number of points of each table is defined by means of the general machine parameter “NPCROSS”, "NPCROSS2" and "NPCROSS3" respectively.

Each table defines:

* The position of the axis causing the error.

* The error suffered by the axis at that point.

Also, the current position of the selected axis is displayed. This position is updated as the axis moves.

11.5 OPERATION WITH PARAMETER TABLES

Once one of the tables has been selected, the cursor can be moved over the screen line by line by means of the “up and down arrow keys” or move from page to page by means of the “page up and page down keys”.

In addition, the user has an area of the screen for editing, it being possible to move the cursor over the screen by means of the “right arrow key and left arrow key”.

The CNC offers the following softkey options for each table:

EDIT

The desired parameter.

When selecting this option, the softkeys will change their color to a white background and they will show the various editing options.

In those tables corresponding to leadscrew and cross compensation, the position values of the axis must be edited as follows:

* Move the axis and when the error is found large enough to be considered, press the softkey corresponding to this axis.

* The CNC will include, in the editing area, the name of the axis followed by the position value corresponding to that point.

  This value can be modified if so desired.

* Press the softkey corresponding to the error and key in its value.

Once the parameter is edited, press **ENTER**. This new parameter will be included in the table and the cursor will be positioned over it. The editing area will be cleared, thus allowing other parameters to be edited.

Press **ESC** to quit this mode.

MODIFY

With this option it is possible to modify the selected parameter.

Before pressing this softkey, the desired parameter must be selected.

When selecting this option, the softkeys will change their color to a white background and they will show the various editing options.

By pressing **ESC**, the information displayed in the editing window (corresponding to the selected parameter) will be cleared. From this point on, a new value can be entered.

To quit this option, first clear the editing window using the **CL** key or the **ESC** key and then press **ESC** again. The selected parameter will not be modified.

Once this modification has concluded, press the **ENTER** key to validate it.

FIND

The beginning or end of the table, or the parameter whose number is indicated by positioning the cursor on the required parameter.

BEGINNING When pressing this softkey, the cursor positions over the first parameter of the table quitting this option.

END When pressing this softkey, the cursor positions over the last parameter of the table quitting this option.

PARAMETER When pressing this softkey, the CNC will request the number of the parameter to be found. Key in that number and press **ENTER**. The cursor will be positioned over the indicated parameter quitting this option.

INITIALIZE

With this option it is possible to reset all the parameters of the selected table to their default values.

These default values are indicated in the chapter corresponding to machine parameters in the installation manual.

LOAD

To load tables stored in the "Memkey Card" (CARD A) or in a peripheral device or PC through the two serial lines (RS232C or RS422).

The transmission begins after pressing the corresponding softkey. When using a serial line, the receptor must be ready before starting the transmission.

To interrupt the transmission, press the [ABORT] softkey.

If the length of the table received does not coincide with the length of the current table, the CNC will acts as follows:

If the table received is shorter than the current one, the received lines are modified and the rest keep their previous values.

If the table received is longer than the current one, the CNC updates all the lines of the current table and when detecting that there is no more room, the CNC issues the corresponding error message.

SAVE

The tables may be saved in the "Memkey Card" (CARD A) or in a peripheral device or PC through the two serial lines (RS232C or RS422).

The transmission begins after pressing the corresponding softkey. When using a serial line, the receptor must be ready before starting the transmission.

To interrupt the transmission, press the [ABORT] softkey.

MM/INCHES

Every time this softkey is pressed, the CNC will change the display format of those parameters affected by these units from millimeters to inches and vice versa.

The lower right-hand window will show the units currently selected.

Note that this change does not affect the general machine parameter "**INCHES**" which indicates the measuring units by default.

12. *DIAGNOSIS*

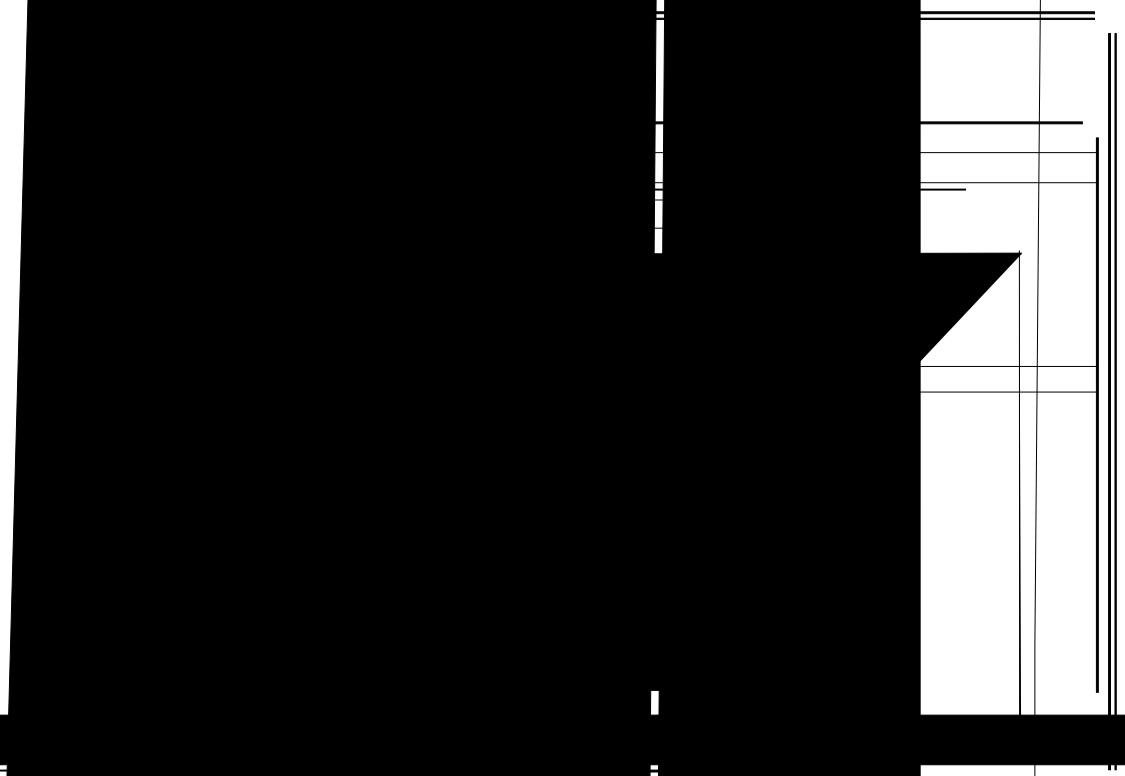
In this operating mode it is possible to know the configuration of the CNC as well as testing the system.

The CNC offers the following softkey options:

System Configuration
Hardware test
Memory test
Flash memory test
Hard Disk

ill appear in order to select the
the system.

e following information:



on of the central unit of the CNC.

the modules and options indicate

CN

It indica e system and for the user.

It also indicates n Kb.

12.1.2 SOFTWARE CONFIGURATION

This option shows:

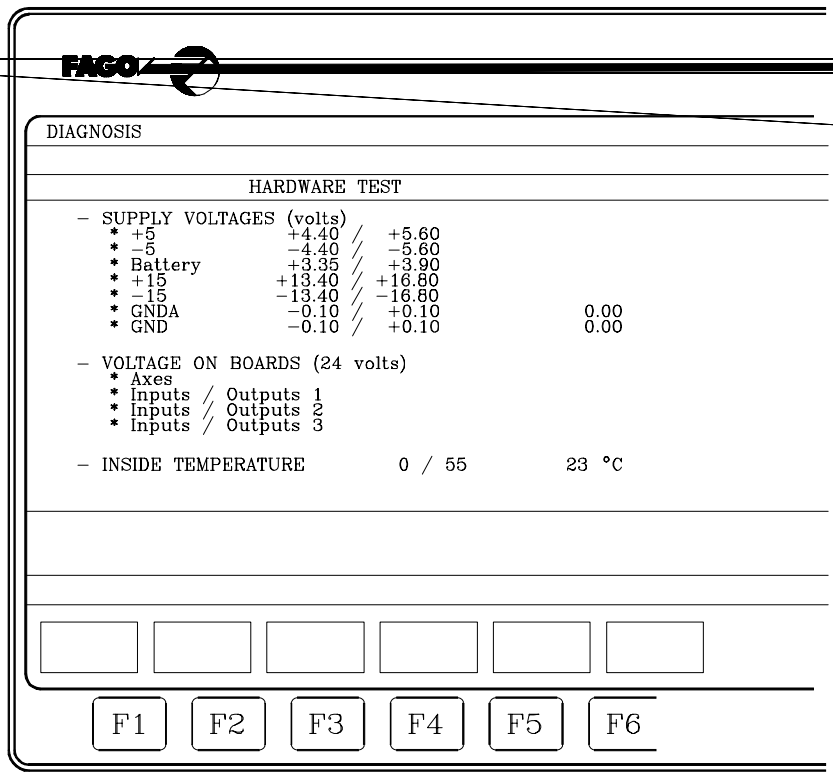
* All available software options.

*

* A ″1ı″id code opf t1ı″unit.

12.2 HARDWARE TEST

This option checks the power supply voltages corresponding to the system and to the boards as well as the internal temperature of the central unit. It displays the following information:



SUPPLY VOLTAGE

It indicates the voltage of the lithium battery and the voltages supplied by the Power Supply Module.

The voltages supplied by the Power Supply module are internally used by the CNC.

Next to the voltages, it displays the value range (maximum and minimum values), the real value and whether it is OK or not.

BOARD VOLTAGE

This section indicates whether the AXES module, the I/O TRACING module and the I/O modules are supplied or not with 24 V.

The lack of these 24V may be because the connectors have not been supplied or because the protection fuse for the corresponding module is blown.

INSIDE TEMPERATURE

It shows the value range (maximum and minimum values), the inside temperature of the Central Unit and whether that value is OK or not.

12.3 MEMORY TEST

This option checks the status of the internal CNC memory, that of the memory available for the User and for the System.

To carry out this verification, the PLC program must be stopped, otherwise, the CNC will ask the operator whether this operation is to be carried out or not.

12.4 FLASH MEMORY TEST

This option checks the status of the internal CNC Flash memory. These memories contain the CNC software version currently installed.

12.5 USER

This option will execute the program which is selected with the general machine parameter "**USERDIAG**" in the user channel.

To quit its execution and return to the previous menu, press **ESC**

12.6 HARD DISK

Once this option has been selected, two softkeys will be displayed:

Test — It check the status of the hard disk (user memory available). It takes about 30 minutes.
In order to perform this test, the PLC program must be stopped. If it is running, the CNC will ask the operator whether it is to be stopped or not.

Compress — It compresses the hard disk by defragmenting it. It also includes a hard disk surface check. The duration of this test depends on the number of files it contains and on how defragmented the hard disk is.

12.7 INTERESTING NOTES

The **CNC** carries out a series of sequential tests.

If the result obtained is not correct, it may stop axes feed and spindle rotation (by cancelling their analog voltages and Enables), as well as stopping the execution of the **PLC** program or activating the external **EMERGENCY** output (**01**).

Test type	**When is it carried out?**	**Stops the axes and the spindle**	**Stops the PLC**	**Activates Emergency output**
Temperature	Always	YES	No	YES
Battery out	Always	No	No	No
Flash memory	After version upgrade	YES	YES	V
(CARD A)	On power-up	No	No	No
RAM memory	On power-up	YES	No	YES
External emergency	EXEC/SIMUL	YES	No	YES
Board voltage	EXEC/SIMUL	YES	No	YES
PLC running	EXEC/SIMUL	YES	---	YES
PLC user error	EXEC/SIMUL	YES	No	No
PLC Watchdog	PLC running	YES	YES	YES

CNC 8055 M

Programming Manual

Ref. 9909 (in)

Please note that the availability of some of the features described in this manual depends on the software options you just obtained.

MODEL	GP	M
Electronic threading	Not available	Available
Tool magazine management	Not available	Available
Solid Graphics	Not available	Option
Machining canned cycles	Not available	Available
Multiple machining	Not available	Available
Probing canned cycles	Not available	Option
Tool life monitoring	Not available	Option
Irregular pockets with islands	Not available	Option
Digitizing	Not available	Option
Tracing	Not available	Option
TCP transformation	Not available	Option
Tool radius compensation	Option	Available
DNC	Option	Option
Software for 7 axes	Option	Option
Profile editor	Option	Option
Rigid tapping	Option	Option
Tangential control	Not available	Option
Conversational Software (MC model)	Not available	Option

---------- o ----------

The information described in this manual may be subject to variations due to technical modifications.

FAGOR AUTOMATION, S.Coop. Ltda. reserves the right to modify the contents of the manual without prior notice.

INDEX

Section page

VERSION HISTORY

INTRODUCCIÓN

Safety conditions 3
Material returning terms 5
Fagor documentation for the CNC 6
Manual contents 7

1. OVERVIEW

1.1 Part-programs 1
1.1.1 Considerations for the Ethernet connection 4
1.2 DNC connection 6
1.3 Communication protocol via dnc or peripheral device 6

2. CREATING A PROGRAM

2.1 Creating a program in the cnc 1
2.1.1 Block header 2
2.1.2 Program block 3
2.1.2.1 Iso language 3
2.1.2.2 High level language 3
2.1.3 End of block 4

3. AXES AND COORDINATE SYSTEMS

3.1 Nomenclature of the axes 1
3.1.1 Selection of the axes 2
3.2 Plane selection (G16, G17, G18, G19) 3
3.3 Part dimensioning. millimeters (G71) or inches (G70) 5
3.4 Absolute/incremental programming (G90, G91) 6
3.5 Programming of coordinates 7
3.5.1 Cartesian coordinates 7
3.5.2 Polar coordinates 8
3.5.3 Cylindrical coordinates 10
3.5.4 Angle and one cartesian coordinate 11
3.6 Rotary axes 12
3.7 Work zones 13
3.7.1 Definition of the work zones 13
3.7.2 Using work zones 14

Section page

4. REFERENCE SYSTEMS

4.1 Reference points 1
4.2 Machine reference search (G74) 2
4.3 Programming with respect to machine zero (G53) 3
4.4 Presetting of coordinates and zero offsets 4
4.4.1 Coordinate preset and limitation of the s value (G92) 6
4.4.2 Zero offsets (G54..G59) 7
4.5 Polar origin preset (G93) 9

5. PROGRAMMING BY ISO CODE

5.1 Preparatory functions 2
5.2 Feedrate functions (G94, G95) 4
5.2.1 Feedrate in mm/min or inches/min (G94) 4
5.2.2 Feedrate in mm/rev.or inches/rev (G95) 5
5.3 Spindle selection (G28, G29) 6
5.4 Constant speed functions (G96,G97) 7
5.4.1 Constant surface speed (G96) 7
5.4.2 Constant tool-center speed (G97) 7
5.5 Complementary functions 8
5.5.1 Feedrate F 8
5.5.2 Spindle speed and spindle orientation (S) 9
5.5.3 Tool number (T) 10
5.5.4 tool offset number (D) 11
5.5.5 Miscellaneous function (M) 12
5.5.5.1 M00. Program stop 13
5.5.5.2 M01. Conditional program stop 13
5.5.5.3 M02. End of program 13
5.5.5.4 M30. End of program with return to first block 13
5.5.5.5 M03. Clockwise spindle rotation 13
5.5.5.6 M04. Counter-clockwise spindle rotation 13
5.5.5.7 M05. Spindle stop 13
5.5.5.8 M06. Tool change 14
5.5.5.9 M19. Spindle orientation 14
5.5.5.10 M41, M42, M43, M44. Spindle speed range change 15
5.5.5.11 M45 auxiliary spindle / live tool 15

6. PATH CONTROL

6.1 Rapid travel (G00) 1
6.2 linear interpolation (G01) 2
6.3 Circular interpolation (G02. G03) 3
6.4 Circular interpolation by programming the center of the arc in absolute coordinates (G06) 9
6.5 Arc tangent to the previous path (G08) 10
6.6 Arc defined by three points (G09) 11
6.7 Helical interpolation 12
6.8 Tangential entry at beginning of a machining operation (G37) 14
6.9 Tangential exit at the end of a machining operation (G38) 16
6.10 Automatic radius blend (G36) 18
6.11 Automatic chamfer blend (G39) 19
6.12 Threading (G33) 20
6.13 Move to hardstop (G52) 21
6.14 Feedrate "F" as an inverted function of time (G32) 22

Section page

6.15 Tangential control (G45) 23
6.15.1 Considerations about function G45 25

7. ADDITIONAL PREPARATORY FUNCTIONS

7.1 Interruption of block preparation (G04) 1
7.2 Dwell (G04 K) 3
7.3 Working with square (G07) and round (G05,G50) corners 4
7.3.1 Square corner (G07) 4
7.3.2 Round corner (G05) 5
7.3.3 Controlled round corner (G50) 6
7.4 Look-ahead (G51) 7
7.5 Mirror image (G10, G11. G12, G13, G14) 9
7.6 Scaling factor (G72) 11
7.6.1 Scaling factor applied to all axes 12
7.6.2 Scaling factor applied to one or more axes 14
7.7 Pattern rotation (G73) 16
7.8 Slaved axis/cancellation of slaved axis 18
7.8.1 Slaved axis (G77) 19
7.8.2 Slaved axis cancellation (G78) 20
7.9 Axes toggle. G28-G29 21

8. TOOL COMPENSATION

8.1 Tool radius compensation (G40, G41, G42) 2
8.1.1 Activating tool radius compensation 3
8.1.2 Tool radius compensation sections 6
8.1.3 Cancelling tool radius compensation 9
8.2 Tool length compensation (G43, G44, G15) 15

9. CANNED CYCLES

9.1 Definition of a canned cycle 1
9.2 Canned cycle area of influence 2
9.2.1 G79. Modification of canned cycle parameters 2
9.3 Canned cycle cancellation 4
9.4 General considerations 5
9.5 Machining canned cycles 6
9.5.1 G69. Complex deep hole drilling cycle 8
9.5.2 G81 drilling canned cycle 12
9.5.3 G82. Drilling canned cycle with dwell 14
9.5.4 G83. Simple deep hole drilling 16
9.5.5 G84. Tapping canned cycle 19
9.5.6 G85. Reaming cycle 22
9.5.7 G86. Boring cycle with withdrawal in rapid (G00) 24
9.5.8 G87. Rectangular pocket canned cycle 26
9.5.9 G88. Circular pocket canned cycle 34
9.5.10 G89. Boring cycle with withdrawal at working feedrate (G01) 41

Section page

10. MULTIPLE MACHINING

10.1 G60: Multiple machining in a straight line pattern 2
10.2 G61: Multiple machining in a rectangular pattern 5
10.3 G62: Multiple machining in a grid pattern 8
10.4 G63: Multiple machining in a circular (bolt-hole) pattern 11
10.5 G64: Multiple machining in an arc pattern 14
10.6 G65: Machining programmed by means of an arc chord 17

11. IRREGULAR POCKET CANNED CYCLE (WITH ISLANDS)

11.1 2D pockets 2
11.1.1 Drilling operation 5
11.1.2 Roughing operation 6
11.1.3 Finishing operation 9
11.1.4 Profile programming rules 12
11.1.5 Intersection of profiles 13
11.1.5.1 Basic profile intersection (k=0) 13
11.1.5.2 Advanced profile intersection (k=1) 14
11.1.5.3 Resulting profile 16
11.1.6 Profile programming syntax 17
11.1.7 Errors 19
11.1.8 Programming examples 21
11.2 3D pockets 25
11.2.1 Roughing operation 29
11.2.2 Semi-finishing operation 32
11.2.3 Finishing operation 34
11.2.4 Profile or contour geometry 36
11.2.5 Profile programming rules 37
11.2.5.1 Programming examples 39
11.2.6 Composite 3d profiles 42
11.2.6.1 Profile intesecting rules 43
11.2.7 Stacked profiles 45
11.2.8 Profile programming syntax 46
11.2.9 Examples 48
11.2.10 Errors 59

12. WORKING WITH A PROBE

12.1 Probing (G75,G76) 2
12.2 Probing canned cycles 3
12.3 Tool length calibration canned cycle 4
12.4 Probe calibrating canned cycle 7
12.5 Surface measuring canned cycle 11
12.6 Outside corner measuring canned cycle 15
12.7 Inside corner measuring canned cycle 18
12.8 Angle measuring canned cycle 21
12.9 Outside corner and angle measuring canned cycle 24
12.10 Hole measuring canned cycle 28
12.11 Boss measuring canned cycle 32

Section page

13. PROGRAMMING IN HIGH-LEVEL LANGUAGE

13.1 Lexical description 1
13.1.1 Reserved words 2
13.1.2 Numerical constants 3
13.1.3. Symbols 3
13.2 Variables 4
13.2.1 General purpose parameters or variables 6
13.2.2 Variables associated with tools 8
13.2.3 Variables associated with zero offsets 10
13.2.4 Variables associated with function g49 11
13.2.5 Variables associated with machine parameters 12
13.2.6 Variables associated with work zones 13
13.2.7 Variables associated with feedrates 14
13.2.8 Variables associated with coordinates 16
13.2.9 Variables associated with the electronic handwheels *17*
13.2.10 Variables associated with the main spindle *18*
13.2.11 Variables associated with the 2nd spindle 20
13.2.12 Variables associated with the plc 22
13.2.13 Variables associated with local parameters 23
13.2.14 Sercos variables 24
13.2.15 Other variables 25
13.3 Constants 32
13.4 Operators *32*
13.5 Expressions 34
13.5.1 Arithmetic expressions 34
13.5.2 Relational expressions 35

14. PROGRAM CONTROL STATEMENTS

14.1 Assignment statements 1
14.2 Display statements 2
14.3 Enabling-disabling statements 3
14.4 Flow control statements 4
14.5 Subroutine statements 6
14.5.1 Interruption subroutine statements 12
14.6 Program statements 13
14.7 Screen customizing statements (graphic editor) 16

15. DIGITIZING CYCLES

15.1 Digitizing cycle in a grid pattern 2
15.2 Digitizing cycle in an arc pattern 5

Section page

16. TRACING AND DIGITIZING

16.1 Introduction 1
16.1.1 General considerations 7
16.2 G26. Calibration of the tracing probe 9
16.3 G23. Activate tracing 11
16.3.1 G23. Activate manual tracing 12
16.3.2 G23. Activate one-dimensional tracing 14
16.3.3 G23. Activate two-dimensional tracing 16
16.3.4 G23. Activate three-dimensional tracing 18
16.4 G27. Tracing contour definition 20
16.5 G25. Deactivate tracing 24
16.6 G24. Activate digitizing 25
16.7 Tracing and digitizing canned cycles 28
16.7.1 Grid -pattern tracing canned cycle 29
16.7.2 Arc pattern tracing canned cycle 34
16.7.3 Profile tracing canned cycle along a plane 40
16.7.4 3D profile tracing canned cycle 45
16.7.5 Tracing canned cycle with polygonal sweep 50
16.7.5.1 Profile programming rules 55
16.7.5.2 Profile programming syntax 56

17. COORDINATE TRANSFORMATION

17.1 Movement in the Incline Plane 7
17.1.1 Incline plane definition (G49) 8
17.1.2 Considerations for function G49 12
17.1.3 Variables associated with function G49 13
17.1.4 Parameters associated with function G49 13
17.1.5 Programming example 14
17.2 Movement according to the tool coordinate system (G47) 15
17.3 TCP Transformation (G48) 16
17.3.1 Considerations for function G48 20

APPENDIX

A. Iso code programming 3
B. Variables associated with tools 5
C. High level programming 10
D. Key codes 12
E. Logic outputs for key code status 13
F. Key inhibiting codes 14
G. Programming assistance system pages 15
H. Maintenance 18

VERSION HISTORY (M)

(MILL MODEL)

Date: **May 1999** ***Software Version:*** **3.0x**

FEATURE	AFFECTED MANUAL & CHAPTERS	
Portuguese language	Installation Manual	Chapter 3
Tangential Control	Installation Manual Programming Manual	Chapters 9, 10, Appendix Chapters 6, 13, Appendix
Incline planes. The software travel limits are monitored in JOG movements.		
PLC. User registers R1 through R499	Installation Manual Programming Manual	Chapters 6, 7, Appendix Chapter 13
CNC status screen	Operation Manual	Chapter 8
Hard disk (HD)	Installation Manual	Chapters 1, 3, Appendix
HD Diagnosis	Operation Manual	Chapter 12
Integrate the HD into an outside PC network	Installation Manual	Chapter 3
Consult directories, delete, rename and copy programs in the same or other device	Operation Manual Programming Manual	Chapters 1, 7 Chapter 1
Ejecution and simulacion from RAM memory, Memkey Card, HD or serial line.	Operation Manual	Chapters 1, 3,
It is possible to execute (EXEC) and open (OPEN) a program (to be edited) stored in any device.	Programming Manual	Chapter 14, Appendix
MC option. Tool calibration screen. When defining R and L; I and K are initialized If I=0 and K=0; I and K are initialized	Operation Manual	Chapter 3
MC option. ISO management, also as MDI	MC Operation Manual	Chapter 3
MC option. New way to handle safety planes.	MC Operation Manual	Chapter 4
MC option. New codes for specific keys.	MC Operation Manual	Appendix

INTRODUCTION

SAFETY CONDITIONS

Read the following safety measures in order to prevent damage to personnel, to this product and to those products connected to it.

This unit must only be repaired by personnel authorized by Fagor Automation.

Fagor Automation shall not be held responsible for any physical or material damage derived from the violation of these basic safety regulations.

Precautions against personal damage

Before powering the unit up, make sure that it is connected to ground

In order to avoid electrical discharges, make sure that all the grounding connections are properly made.

Do not work in humid environments

In order to avoid electrical discharges, always work under 90% of relative humidity (non-condensing) and 45º C (113º F).

Do not work in explosive environments

In order to avoid risks, damage, do no work in explosive environments.

Precautions against product damage

Working environment

This unit is ready to be used in Industrial Environments complying with the directives and regulations effective in the European Community

Fagor Automation shall not be held responsible for any damage suffered or caused when installed in other environments (residential or homes).

Install the unit in the right place

It is recommended, whenever possible, to instal the CNC away from coolants, chemical product, blows, etc. that could damage it.

This unit complies with the European directives on electromagnetic compatibility. Nevertheless, it is recommended to keep it away from sources of electromagnetic disturbance such as.

- Powerful loads connected to the same AC power line as this equipment.
- Nearby portable transmitters (Radio-telephones, Ham radio transmitters).
- Nearby radio / TC transmitters.
- Nearby arc welding machines
- Nearby High Voltage power lines
- Etc.

Ambient conditions

The working temperature must be between +5° C and +45° C (41ºF and 113º F)
The storage temperature must be between -25° C and 70° C. (-13º F and 158º F)

Protections of the unit itself

Power Supply Module

It carries two fast fuses of 3.15 Amp./ 250V. to protect the mains AC input

Axes module

All the digital inputs and outputs have galvanic isolation via optocouplers between the CNC circuitry and the outside.
They are protected by an external fast fuse (F) of 3.15 Amp./ 250V. against reverse connection of the power supply.

Input / Output Module

All the digital inputs and outputs have galvanic isolation via optocouplers between the CNC circuitry and the outside.
They are protected by an external fast fuse (F) of 3.15 Amp./ 250V. against a voltage overload (greater than 33Vdc) and against reverse connection of the power supply.

Input / Output and Tracing Module

All the digital inputs and outputs have galvanic isolation via optocouplers between the CNC circuitry and the outside.
They are protected by an external fast fuse (F) of 3.15 Amp./ 250V. against a voltage overload (greater than 33Vdc) and against reverse connection of the power supply.

Fan Module

It carries 1 or 2 external fuses depending on model
The fuses are fast (F), of 0.4 Amp./ 250V. to protect the fans.

Monitor

The type of protection fuse depends on the type of monitor. See the identification label of the unit itself.

Precautions during repair

Do not manipulate the inside of the unit

Only personnel authorized by Fagor Automation may manipulate the inside of this unit.

Do not manipulate the connectors with the unit connected to AC power.

Before manipulating the connectors (inputs/outputs, feedback, etc.) make sure that the unit is not connected to AC power.

Safety symbols

Symbols which may appear on the manual

WARNING. symbol
It has an associated text indicating those actions or operations may hurt people or damage products.

Symbols that may be carried on the product

WARNING. symbol
It has an associated text indicating those actions or operations may hurt people or damage products.

"Electrical Shock" symbol
It indicates that point may be under electrical voltage

"Ground Protection" symbol
It indicates that point must be connected to the main ground point of the machine as protection for people and units.

MATERIAL RETURNING TERMS

When returning the Monitor or the Central Unit, pack it in its original package and with its original packaging material. If not available, pack it as follows:

1.- Get a cardboard box whose three inside dimensions are at least 15 cm (6 inches) larger than those of the unit. The cardboard being used to make the box must have a resistance of 170 Kg (375 lb.).

2.- When sending it to a Fagor Automation office for repair, attach a label indicating the owner of the unit, person to contact, type of unit, serial number, symptom and a brief description of the problem.

3.- Wrap the unit in a polyethylene roll or similar material to protect it.

When sending the monitor, especially protect the CRT glass

4.- Pad the unit inside the cardboard box with poly-utherane foam on all sides.

5.- Seal the cardboard box with packing tape or industrial staples.

FAGOR DOCUMENTATION FOR THE CNC

OEM Manual	Is directed to the machine builder or person in charge of installing and starting-up the CNC.
USER Manual	Is directed to the end user or CNC operator. It contains 2 manuals: Operating Manual — describing how to operate the CNC. Programming Manual — describing how to program the CNC.
DNC Software Manual	Is directed to people using the optional DNC communications software.
DNC Protocol Manual	Is directed to people wishing to design their own DNC communications software to communicate with the CNC.
FLOPPY DISK Manual	Is directed to people using the Fagor Floppy Disk Unit and it shows how to use it.

MANUAL CONTENTS

The Programming Manual for the Mill model CNC contains the following chapters:

Index

New Features and Modifications for the Mill Model

Introduction
- Summary of safety conditions
- Shipping terms
- Fagor documentation for the CNC.
- Manual contents

Chapter 1 Overview
- It shows how to enter part-programs from the keyboard or via DNC.
- Location of part-programs, how to edit them and execute them.
- It indicates the protocol to be used in DNC communications.

Chapter 2 Creating a program
- It indicates the structure for a part-program and all its blocks.
- It shows the languages that could be used to program the parts: ISO coded and High-Level languages

Chapter 3 Axes and coordinate systems
- It indicates the nomenclature of the axes and how to select them.
- It shows how to select the working planes, work units, type of programming system (absolute /incremental).
- It describes the coordinates systems that could be used for programming: Cartesian, polar, cylindric, angle plus Cartesian coordinate.
- It shows how to operate with rotary axes and how to define and use work zones.

Chapter 4 Reference systems
- It indicates the machine reference (home) and datum points to be set at the CNC.
- It shows how to program a home search, how to program coordinates with respect to home, how to preset coordinates, zero offsets and polar origins.

Chapter 5 Programming by ISO code
- It shows how to program preparatory functions for feedrate and constant speed as well as additional functions such as "F, S, T, D and M".
- It describes how to select the main spindle or the auxiliary spindle

Chapter 6 Path control
- It shows how to program rapid traverse, linear, circular and helical interpolations.
- It shows how to program tangential entries and exits as well as corner rounding and chamfering
- It shows how to program electronic threading and movements against hard stop.
- It shows how to program the feedrate as an inverted function of time.
- It shows how to program the tangential control

Chapter 7 Additional preparatory functions
- It shows how to interrupt block preparation and how to program a dwell.
- It shows how to program a part in square corner, round corner or with an automatic radius blend.
- It describes how to program the look-ahead, mirror image, scaling factor, pattern rotation and the electronic slaving / unslaving of the axes.

Chapter 8 Tool compensation
- It shows how to program tool radius and length compensation.

Chapter 9 Canned cycles
- It shows how to program the different machining canned cycles.

Chapter 10 Multiple machining

It shows how to program the different multiple machining cycles.

Chapter 11 Irregular pocket canned cycles (with islands)

It shows how to program the different 2-D and 3-D pocket canned cycles.

Chapter 12 Working with a probe

It shows how to carry out probing moves and how to program the probing canned cycles.

Chapter 13 Programming in high level language

It shows all the variables, symbols, operators, etc. to be used when programming in high level language.

Chapter 14 Program control statements

It shows the control sequences that can be used in high-level language. The available instructions are: for assignment, display enable/disable, flow control, subroutines and for generating programs and screens.

Chapter 15 Digitizing cycles

It shows how to program the various digitizing cycles.

Chapter 16 Tracing and Digitizing

It shows how to program the various digitizing and tracing cycles.

Chapter 17 Coordinate transformation

It describes coordinate transformation.
It shows how to select incline planes.
It shows how to make movements along the tool axes.
It shows how to work with TCP (Tool Center Point) transformation.

Appendix
A ISO code programming
B Internal CNC variables
C High level programming
D Key codes
E Programming assistance system pages

1. OVERVIEW

The CNC can be programmed both at the machine (from the front panel) or from external peripheral devices (tape reader/cassette recorder, computer, etc. Memory available to the user for carrying out the part programs is 1 Mbyte.

The part programs and the values in the tables which the CNC has can be entered as follows :

* **From the front panel**. Once the editing mode or table required has been selected, the CNC allows you to enter data from the keyboard.

* **From a Computer (DNC) or Peripheral Device**. The CNC allows data to be interchanged with a computer or peripheral device, using RS232C and RS422 cables.

  If this is controlled from the CNC, it is necessary to preset the corresponding table or part program directory (utilities) you want to communicate with.

  Depending on the type of communication required, the serial port machine parameter "PROTOCOL" should be selected.

  "PROTOCOL" = 0 if the communication is with a peripheral device.
  "PROTOCOL" = 1 if the communication is via DNC.

1.1 PART-PROGRAMS

Editing

To create a part-program, access the Edit mode. See chapter 5 in this manual.

The new part-program edited is stored in the CNC's RAM memory.

A copy of the part-programs may be stored in the "MemKey Card", at a PC connected through serial line 1 or 2 or in the hard disk (HD module). See chapter 7 in this manual.

When using a PC through serial line 1 or 2, proceed as follows:
- Execute the "Fagor50.exe" applications program at the PC.
- Activate DNC communications at the CNC. See chapter 8 in this manual.
- Select the work directory as shown in chapter 7 of this manual. Option: Utilities\ Directory\ Serial L.\ Change directory.

With the Edit mode of operation, part-programs residing in the CNC's RAM memory may be modified. To modify a program stored in the "MemKey Card", in a PC or in the hard disk, it must be previously copied into RAM memory.

Execution

Part-programs stored anywhere may be executed or simulated. See chapter 3 in this manual.

The user customizing programs must be in RAM memory so the CNC can execute them.

The GOTO and RPT instructions cannot be used in programs executed from a PC connected through the serial lines. See chapter 14 of the programming manual.

The subroutines can only be executed if they reside in the CNC's RAM memory. Therefore, to execute a subroutine stored in the "MemKey Card", in a PC or in the hard disk, it must be first copied into the CNC's RAM memory.

From a program in execution, another program can be executed which is in RAM memory, in the "MemKey Card", in a PC or in the hard disk using the EXEC instruction. See chapter 14 of the programming manual.

Utilities

This operating mode, chapter 7 of this manual, lets display the part-program directory of all the devices, make copies, delete, rename and even set the protections for any of them.

Ethernet

When having the Ethernet option and if the CNC is configured as another node within the computer network, the following operations are possible from any PC of the network:

- Access the part-program directory of the Hard Disk(HD).
- Edit, modify, delete, rename, etc.the programs stored on the hard disk (HD).
- Copy programs from the hard disk to the PC and vice versa.

To configure the CNC as another node within the computer network, see section 3.3.4 of the installation manual.

Operations that may be carried out with part-programs:

	RAM Memory	CARD A	HD	DNC
Consult the program directory in ...	Yes	Yes	Yes	Yes
Consult the subroutine directory in ...	Yes	No	No	No
Create work directory in ..	No	No	No	No
Change work directory in ..	No	No	No	Yes
Edit a program in ..	Yes	No	No	No
Modify a program in ..	Yes	No	No	No
Delete a program from ..	Yes	Yes	Yes	Yes
Copy from/to RAM memory to/from ...	Yes	Yes	Yes	Yes
Copy from/to CARD A to/from ...	Yes	Yes	Yes	Yes
Copy from/to HD to/from ...	Yes	Yes	Yes	Yes
Copy from/to DNC to/from ...	Yes	Yes	Yes	Yes
Rename a program in ..	Yes	Yes	Yes	No
Change the comment of a program in ..	Yes	Yes	Yes	No
Change protections of a program in ..	Yes	Yes	Yes	No
Execute a part-program in ..	Yes	Yes	Yes	Yes
Execute a user program in ..	Yes	No	No	No
Execute the PLC program in ..	Yes	*	No	No
Execute programs using the GOTO or RPT instructions from ..	Yes	Yes	Yes	No
Execute subroutines stored in ..	Yes	No	No	No
Execute programs stored in RAM, CARD A or HD using the EXEC instruction from ..	Yes	Yes	Yes	Yes
Execute programs via DNC with the EXEC instruction from ..	Yes	Yes	Yes	No
Open programs stored in RAM, CARD A or HD using the OPEN instruction from ..	Yes	Yes	Yes	Yes
Open programs via DNC using the OPEN instruction from ..	Yes	Yes	Yes	No
Consult from a PC and through Ethernet, the program directory in ...	No	No	Yes	No
Consult from a PC and through Ethernet, the subroutine directory in ...	No	No	No	No
Create from a PC and through Ethernet, a directory in...	No	No	No	No

* If it is not in RAM memory, it generates an executable code in RAM and it executes it..

1.1.1 CONSIDERATIONS FOR THE ETHERNET CONNECTION

When configuring the CNC as another node in the computer network , the programs stored in the hard disk module (HD) may be edited and modified from any PC.

Instructions for seting up a user PC to access CNC directories

Recommended configuration:

- Open the «Windows Explorer»
- On the «Tools» menu, select the «Connect to Network Drives» option.
- Select the Drive. For example: «D»
- Indicate the path: CNC name followed by the name of the shared directory. For example: \\FAGOR8055\CNCHD
- When selecting the option: «Connect again when initiating the session», the selected CNC will appear on each power-up as another path of the «Windows Explorer» without having to define it again.

This connection is established through Ethernet and, therefore, the CNC does not control the syntax of the programs while they are received or modified.

Whenever accessing the program directory of the Hard Disk (HD), the following verification takes place:

File name.

The file number must always have 6 digits and the extension PIM (for milling) or PIT (for lathe). Examples: 001204.PIM 000100.PIT

If the file has been given the wrong name, for example: 1204.PIM or 100.PIT, the CNC will not change it, but it will display it with the comment: '**********'.

The file cannot be modified from the CNC. It must be edited from the PC to correct the error.

File size.

If the file is empty, size = 0, the CNC will display it with the comment: '**********'.

The file can be edited or deleted either from the CNC or from the PC.

First line of the program

The first line of the program must have the % character, the comment associated with the file (up to 20 characters) and between the 2 commas (,) the program attributes: O (OEM), H (hidden), M (modifiable), X (executable).

Examples %Comment ,MX,
% ,OMX,

If the first line does not exist. The CNC will display it with an empty comment and with the modifiable (M) and executable (X) attributes.

When the format of the first line is wrong, the CNC does not modify it, but it displays it with the comment: '**********'. The file can be modified or deleted from the CNC or from the PC.

It is the wrong format when:
- the comment has more than 20 characters
- a comma (,) is missing for separating the attributes
- the attributes have a strange character

1.2 DNC CONNECTION

The CNC offers as optional feature the possibility of working in DNC (Distributed Numerical Control), enabling communication between the CNC and a computer to carry out the following functions :

* Directory and delete commands.
* Transfer of programs and tables between the CNC and a computer.
* Remote control of the machine.
* The ability to supervise the status of advanced DNC systems.

1.3 COMMUNICATION PROTOCOL VIA DNC OR PERIPHERAL DEVICE

This type of communication enables program-and-table transfer commands, plus the organization of CNC directories such as the Computer Directory, for copying/deleting programs, etc. to be done either from the CNC or the computer.

When you want to transfer files, it is necessary to follow this protocol :

* The "%" symbol will be used to start the file, followed by the **program comment** (optional), of up to 20 characters.

  Then, and separated by a comma ",", comes the attribute (protection) each file has: reading, modifying, etc. This protection is optional and does not have to programmed.

  To end the file header, **RETURN (RT)** or **LINE FEED (LF)** characters should be sent separated by a comma (",").

  Example :

  ```
  %Fagor Automation, -MX, RT
  ```

* Following the header, the file blocks should be programmed. These will all be programmed according to the programming rules indicated in this manual. After each block, to separate it from the others, the **RETURN (RT)** or **LINE FEED (LF)** characters should be used.

  Example :

  ```
  N20 G90 G01 X100 Y200 F2000 LF
  (RPT N10, N20) N3 LF
  ```

If communication is made with a peripheral device, you will need to send the 'end of file' command. This command is selected via the machine parameter for the serial port: "EOFCHR", and can be one of the following characters :

ESC ESCAPE
EOT END OF TRANSMISSION
SUB SUBSTITUTE
EXT END OF TRANSMISSION.

2. *CREATING A PROGRAM*

A CNC (numerical control) program consists of a series of blocks or instructions.

These blocks or instructions are made of words composed of capital letters and numerical format.

The CNC's numerical format consists of :

- the symbols **. + -**
- the figures **0 1 2 3 4 5 6 7 8 9**

Programming allows spaces between letters, numbers and symbols, in addition to ignoring the numerical format if it has zero value, or a symbol if it is positive.

The numerical format of a word can be replaced by an arithmetic parameter in programming. Later and during basic execution, the control will replace the arithmetic parameter by its value, for example :

If XP3 has been programmed, during execution the CNC will replace P3 by its numerical value, obtaining results such as X20, X20.567, X-0.003, etc.

2.1 CREATING A PROGRAM IN THE CNC

All the blocks which make up the program have the following structure :

Block header + program block + end of block

2.1.1 BLOCK HEADER

The block header is optional, and may consist of one or more **block skip conditions** and by the **block number or label**. Both can be programmed in this order.

CONDITION FOR BLOCK SKIP, /, /1, /2, /3.

These three block skip conditions, given that "/" and "/1" is the same, are governed by the marks BLKSKIP1, BLKSKIP2 and BLKSKIP3 of the PLC.

If any of these marks is active, the CNC will not execute the block or blocks in which it has been programmed. The execution takes place in the following block.

Up to 3 skip conditions can be programmed in one block. These will be evaluated one by one, respecting the order in which they have been programmed.

The control reads 20 blocks ahead of the one being executed in order to calculate in advance the path to be run.

The condition for block skip will be analyzed at the time when the block is read i.e. 20 blocks before execution.

If the block skip needs to be analyzed at the time of execution, it is necessary to interrupt the block preparation, by programming G4 in the previous block.

BLOCK LABEL OR NUMBER N(0-9999)

This is used to identify the block, and is only used when block references or jumps are made.

They are represented by the letter N followed by up to 4 figures (0-9999). It is not necessary to follow any order, and randomly arranged numbers are allowed.

If two or more blocks with the same label number are present in the same program, the CNC will always give priority to the first number.

Although it is not necessary to program it, by using a SOFTKEY the CNC allows the automatic programming of labels. The programmer can select the initial number and the step between labels.

2.1.2 PROGRAM BLOCK

This is written with commands in ISO and High Level languages.

To prepare a program, blocks written in both languages will be used, although each one should be edited with commands in just one language.

2.1.2.1 ISO LANGUAGE

This language is specially designed to control axis movement, as it gives information and movement conditions, in addition to data on feedrate. It includes :

* Preparatory functions for movement, used to determine geometry and working conditions, such as linear and circular interpolations, threading, etc.
* Control functions for axis feedrate and spindle speeds.
* Tool control functions.
* Complementary functions, with technological instructions.

2.1.2.2 HIGH LEVEL LANGUAGE

This enables access to general purpose variables and to system tables and variables.

It gives the user a number of control sentences which are similar to the terminology used in other languages, such as : IF, GOTO, CALL, etc.

It also allows the use of any type of expression (arithmetic, referential, or logical).

It also has instructions for the construction of loops, plus subroutines with local variables. “Local variable” is understood to mean one which is only recognized by the subroutine in which it has been defined.

It is also possible to create libraries, grouping subroutines with useful and tested functions, which can be accessed from any program.

2.1.3 END OF BLOCK

The end of block is optional and may consist of the indication of **number of repetitions of the block** and of the **block comment**. Both must be programmed in this order.

NUMBER OF REPETITIONS OF THE BLOCK, N(0-9999)

This indicates the number of times the block will be executed.

Movement blocks can only be repeated which, at the time of their execution, are under the influence of a modal subroutine.

In these cases, the CNC executes the programmed move and the active machining operation (canned cycle or modal subroutine) the indicated number of times.

The number of repetitions is represented by the letter N followed by up to 4 digits (0-9999).

The active machining operation does not take place if N0 is programmed. Only the movement programmed within the block takes place.

BLOCK COMMENT

The CNC allows you to incorporate any kind of information into all blocks in the form of a comment.

The comment is programmed at the end of the block, and should begin with the character "**;**".

If a block begins with ";", all its contents will be considered as a comment, and it will not be executed.

Empty blocks are not permitted. They should contain at least one comment.

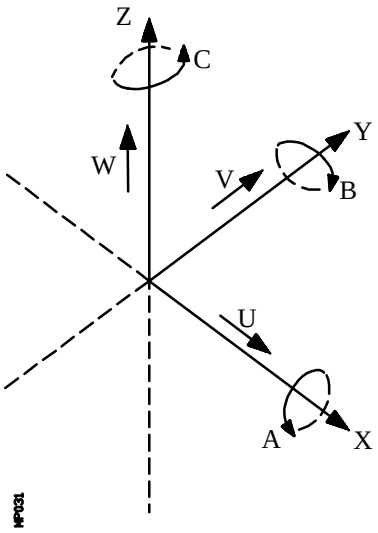
3. *AXES AND COORDINATE SYSTEMS*

Given that the objective of the CNC is to control the movement and positioning of axes, it is necessary to determine, by means of coordinates, the position of the point to be reached.

The CNC allows you to use absolute, relative or incremental coordinates throughout the same program.

3.1 NOMENCLATURE OF THE AXES

The axes are named according to DIN 66217.

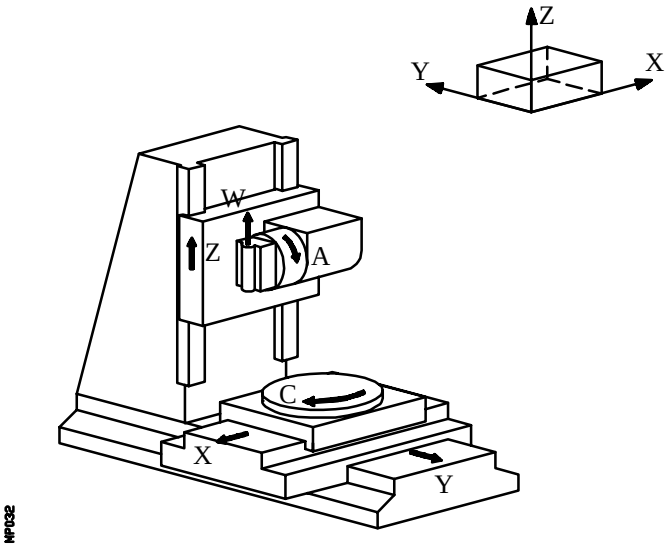


Characteristics of the system of axes :

* **X & Y**: main movements on the main work plane of the machine.
* **Z**: parallel to the main axis of the machine, perpendicular to the main XY plane.
* **U,V,W**: auxiliary axes parallel to X,Y, Z respectively
* **A,B,C**: rotary axes on each of the X,Y, Z axes.

The drawing below shows an example of the nomenclature of the axes on a milling-

profiling machine with a tilted table.



3.1.1 SELECTION OF THE AXES

Of the 9 possible axes which can exist, the CNC allows the manufacturer to select up to 7 of them.

Moreover, all the axes should be suitably defined as linear/rotary, etc. through the axis machine parameters which appear in the Installation and Start-up Manual.

There is no limitation to the programming of the axes, and interpolations can be made simultaneously with up to 7 axes.

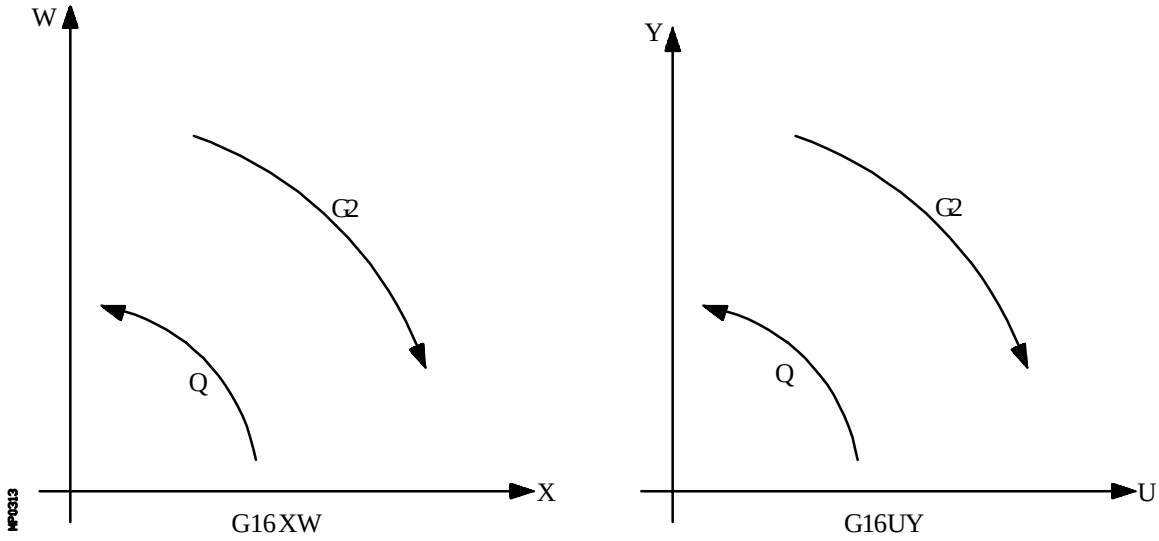
3.2 PLANE SELECTION (G16, G17, G18, G19)

Plane selection should be made when the following are carried out :

- Circular interpolations.
- Controlled corner rounding.
- Tangential entry and exit.
- Chamfer blend.
- Machining canned cycles.
- Pattern rotation.
- Tool radius Compensation.
- Tool length compensation.

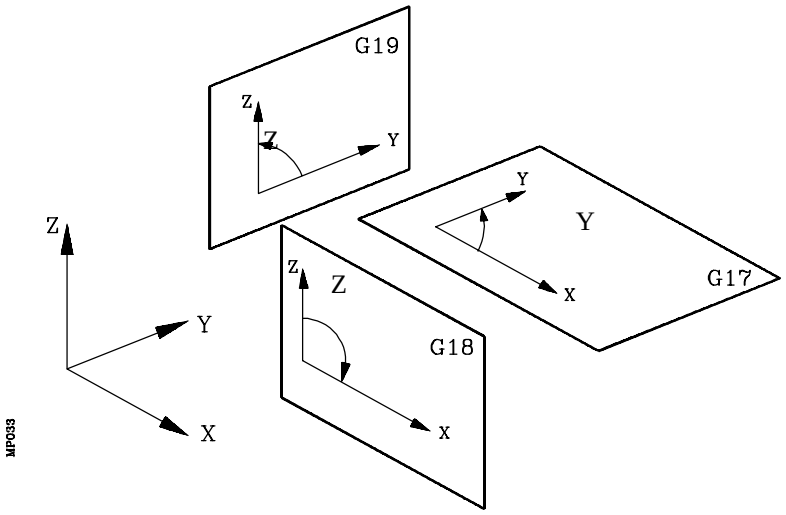
The “G” functions which enable selection of work planes are as follows :

* **G16 axis1 axis2**. Enables selection of the desired work plane, plus the direction of G02 G03 (circular interpolation), **axis1** being programmed as the abscissa axis and **axis2** as the ordinate axis.



* **G17**. Selects the XY plane
* **G18**. Selects the ZX plane
* **G19**. Selects the YZ plane

The G16, G17, G18 and G19 functions are modal and incompatible among themselves. The G16 function should be programmed on its own within a block.



The G17, G18, and G19 functions define two of the three main axes (X, Y, Z) as belonging to the work plane, and the other as the perpendicular axis to the same.

When radius compensation is done on the work plane, and length compensation on the perpendicular axis, the CNC does not allow functions G17, G18, and G19 if any one of the X, Y, or Z axes is not selected as being controlled by the CNC.

On power-up, after executing M02, M30 or after EMERGENCY or RESET, the CNC will assume that the plane defined by the general machine parameter as “IPLANE” is the work plane.

Note: To machine incline planes, function G49 must be used, coordinate transformation, See chapter 17 "Incline planes" on this manual.

3.3 PART DIMENSIONING. MILLIMETERS (G71) OR INCHES (G70)

The CNC allows you to enter units of measurement with the programming, either in millimeters or inches.

It has a general machine parameter “INCHES” to define the unit of measurement of the CNC.

However, these units of measurement can be changed at any time in the program. Two functions are supplied for this purpose :

* G70 : Programming in inches.
* G71 : Programming in millimeters.

Depending on whether G70 or G71 has been programmed, the CNC assumes the corresponding set of units for all the blocks programmed from that moment on.

The G70 and G71 functions are modal and are incompatible.

The CNC allows the programming of figures from 0.0001 to 99999.9999 (with or without sign) when it works in millimeters (G71). This is called format +/- 5.4, or from 0.00001 to 3937.00787 (with or without sign) if it is programmed in inches (G70). This is called format +/- 4.5.

However, and to simplify the instructions, we can say that the CNC admits +/- 5.5 format, thereby admitting +/- 5.4 in millimeters and +/- 4.5 in inches.

On power-up, after executing M02, M30 or after EMERGENCY or RESET, the CNC will assume that the system of units of measurement is the one defined by the general machine parameter “INCHES”.

3.4 ABSOLUTE/INCREMENTAL PROGRAMMING (G90, G91)

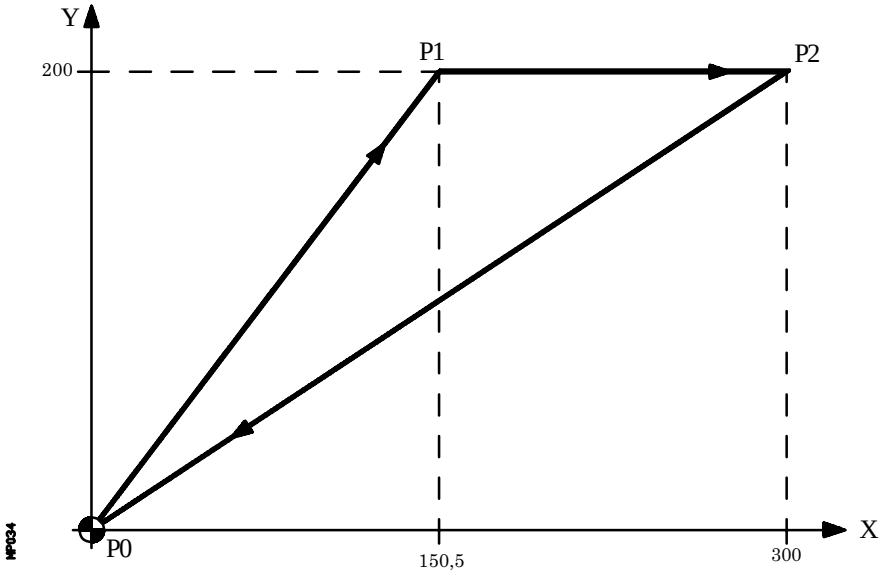
The CNC allows the programming of the coordinates of one point either with absolute **G90** or incremental **G91** values.

When working with absolute coordinates (G90), the point coordinates refer to a point of origin of established coordinates, often the part zero (datum).

When working in incremental coordinates (G91), the numerical value programmed corresponds to the movement information for the distance to be travelled from the point where the tool is situated at that time. The sign in front shows the direction of movement.

Functions G90/G91 are modal and incompatible.

Example :



Absolute coordinates

```
G90  XO      YO     ;  Point P0
     X150.5  Y200   ;  Point P1
     X300           ;  Point P2
     X0      Y0     ;  Point P0
```

Incremental coordinates

```
G90  X0      Y0     ;  Point P0
G91  X150.5  Y200   ;  Point P1
     X149.5         ;  Point P2
     X-300   Y-200  ;  Point P0
```

On power-up, after executing M02, M30 or after an EMERGENCY or RESET, the CNC will assume G90 or G91 according to the definition by the general machine parameter “ISYSTEM”.

3.5 PROGRAMMING OF COORDINATES

The CNC allows the selection of up to 7 of the 9 possible axes X,Y,Z,U,V,W,A,B,C.

Each of these may be linear, linear to position only, normal rotary, rotary to position only or rotary with hirth toothing (positioning in complete degrees), according to the specification in the machine parameter of each “AXISTYPE” axis.

With the aim of always selecting the most suitable coordinate programming system, the CNC has the following types :

* Cartesian coordinates
* Polar coordinates
* Cylindrical coordinates
* Angle and one Cartesian coordinate.

3.5.1 CARTESIAN COORDINATES

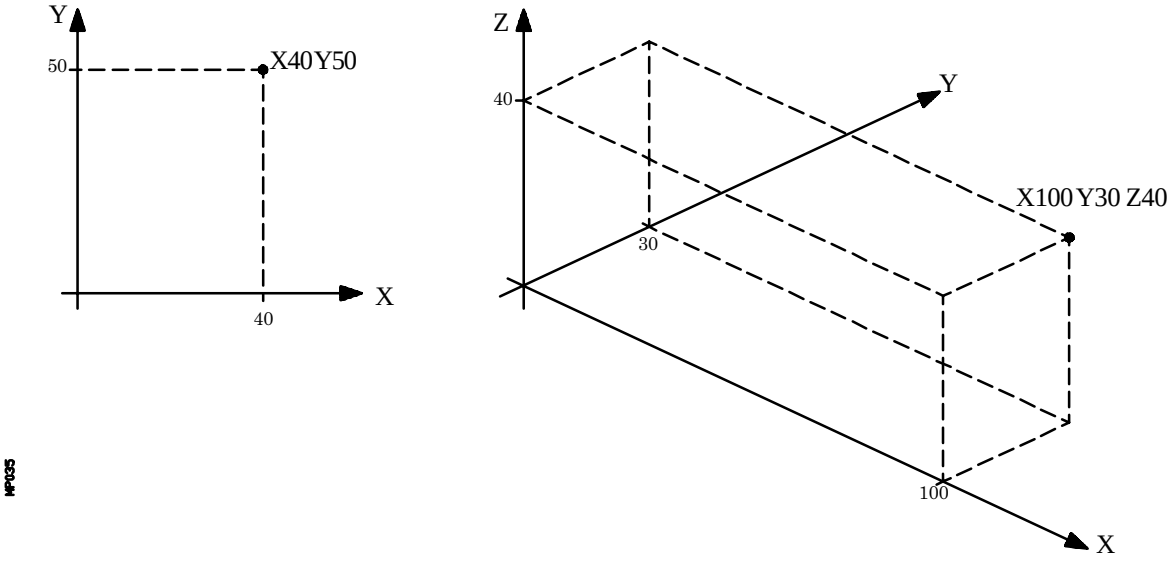
The Cartesian Coordinate System is defined by two axes on the plane, and by three or more axes in space.

The origin of all these, which in the case of the axes X Y Z coincides with the point of intersection, is called **Cartesian Origin** or **Zero Point of the Coordinate System**.

The position of the different points of the machine is expressed in terms of the coordinates of the axes, with two, three, four, or five coordinates.

The coordinates of the axes are programmed via the letter of the axis (X,Y,Z,U,V,W,A,B,C, always in this order) followed by the coordinate value.

The values of the coordinates are absolute or incremental, depending on whether it is working in G90 or G91, and its programming format is +/- 5.5.

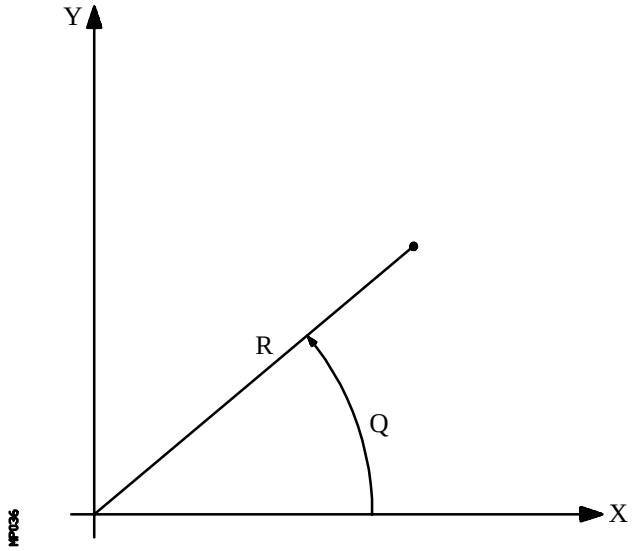


3.5.2 POLAR COORDINATES

In the event of the presence of circular elements or angular dimensions, the coordinates of the different points on the plane (2 axes at the same time), it may be easier to express them in polar coordinates.

The reference point is called **Polar Origin**, and this will be the origin of the **Polar Coordinate System**.

A point on this system would be defined by :



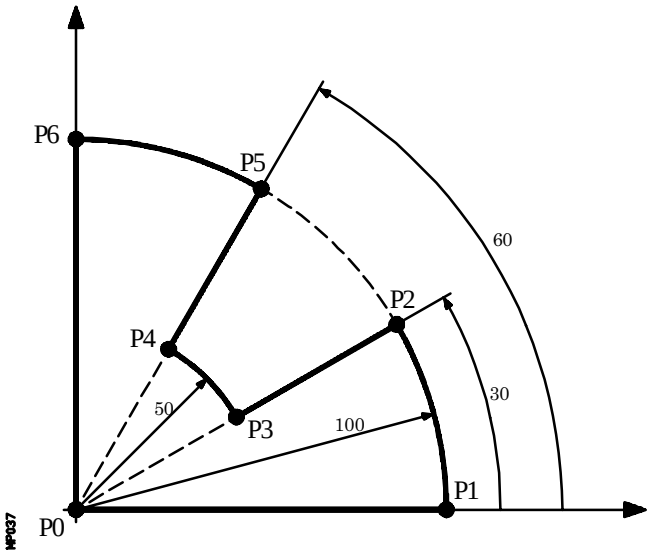
- The **RADIUS (R)**, the distance between the polar origin and the point.
- The **ANGLE (Q)**, formed by the abscissa axis and the line which joins the polar origin with the point (in degrees).

The values **R** and **Q** are absolute or incremental depending on whether you are working with G90 or G91, and their programming format will be **R +/- 5.5 Q+/- 5.5**.

The **R** values **may be negative** when programming in incremental coordinates; but the **resulting** value assigned to the **radius must always be positive**.

If a **Q** value over 3600 is programmed, the module will be taken after dividing it by 360. Thus, Q420 is the same a Q60, and Q-240 is the same as Q-60.

Programming example assuming that the Polar Origin is located at the Coordinate Origin.



Absolute coordinates

```
G90        XO    YO  ; Point P0
     G01 R100  Q0   ; Point P1, in a straight line (G01)
     G03       Q30 ; Point P2, in an arc (G03)
     G01 R50   Q30 ; Point P3, in a straight line (G01)
     G03       Q60 ; Point P4, in an arc (G03)
     G01 R100  Q60 ; Point P5, in a straight line (G01)
     G03       Q90 ; Point P6, in an arc (G03)
     G01 R0    Q90 ; Point P0, in a straight line (G01)
```

Incremental coordinates

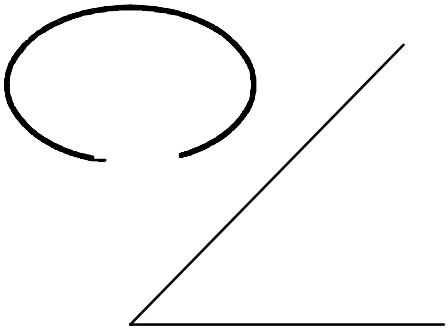
```
G90        X0     Y0  ; Point P0
G91  G01 R100  Q0   ; Point P1, in a straight line (G01)
     G03       Q30 ; Point P2, in an arc (G03)
     G01 R-50  Q0   ; Point P3, in a straight line (G01)
     G03       Q30 ; Point P4, in an arc (G03)
     G01 R50   Q0   ; Point P5, in a straight line (G01)
     G03       Q30 ; Point P6, in an arc (G03)
     G01 R-100 Q0   ; Point P0, in a straight line (G01)
```

The polar origin, apart from being able to be preset using function G93 (described later) can be modified in the following cases :

* On power-up, after executing M02, M30 EMERGENCY or RESET, the CNC will assume, as the polar origin, the coordinate origin of the work plane defined by the general machine parameter"IPLANE".

* Every time the work plane is changed (G16,G17,G18 or G19), the CNC assumes the coordinate origin of the new work plane selected as the polar origin.

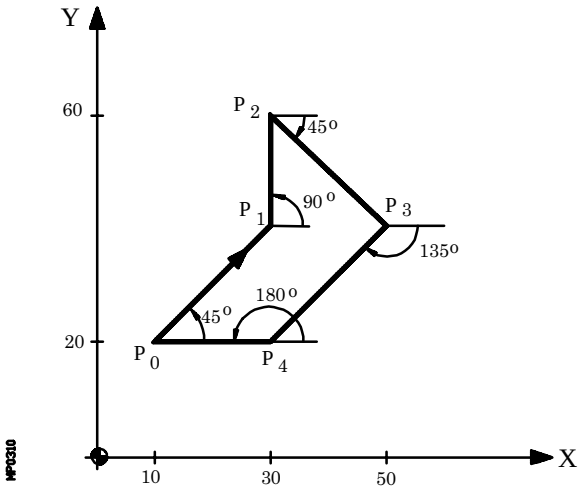
* When executing a circular interpolation (G02 or G03), and if the general machine parameter "PORGMOVE" has a value of 1, the center of the arc will become the new polar origin.



3.5.4 ANGLE AND ONE CARTESIAN COORDINATE

A point on the main plane can be defined via one of its cartesian coordinates, and the exit angle of the previous path.

Example of programming assuming that the main plane is XY:




```
X10    Y20  ;  Point P0, starting point
Q45    X30  ;  Point P1
Q90    Y60  ;  Point P2
Q-45   X50  ;  Point P3
Q-135  Y20  ;  Point P4
Q180   X10  ;  Point P0
```

If you wish to represent a point in space, the remaining coordinates can be programmed in cartesian coordinates.

3.6 ROTARY AXES

The types of rotary axes available are:

Normal rotary axis.
Positioning-only rotary axis.
Hirth rotary axis.

Each one of them can be divided into:
Rollover When their position reading goes from 0º to 360º.
No rollover When their position reading goes from -99999º to 99999º

They are all programmed in degrees. Therefore, their readings are not affected by the inch/mm conversion.

Normal rotary axes

They can be interpolated with linear axes.
Movement: in G00 and G01
Rollover axis programming:
G90 The sign indicates the turning direction and the target position (between 0 and 359.9999).
G91 The sign indicates the turning direction. If the programmed movement exceeds 360º, the axis will rotate more than one turn before positioning at the desired point.
Non-rollover axis programming: In G90 and G91 like a linear axis.

Positioning-only Axes
They cannot be interpolated with linear axes.
Movement: Always in G00 and they do not admit tool radius compensation (G41, G42).
Rollover axis programming:
G90 Always positive and via the shortest path. End coordinate between 0 & 359.9999
G91 The sign indicates the turning direction. If the programmed movement exceeds 360º, the axis will rotate more than one turn before positioning at the desired point.
Non-rollover axis programming: In G90 and G91 like a linear axis.

HIRTH axes

They work like the positioning-only axis except that they do not admit decimal position values (coordinates).

More than one hirth axis can be used, but they can only be moved one at a time.

3.7 WORK ZONES

The CNC provides four work zones or areas, and also limits the tool movement in each of these.

3.7.1 DEFINITION OF THE WORK ZONES

Within each work zone, the CNC allows you to limit the movement of the tool on each axis, with upper and lower limits being defined in each axis.

G20: Defines the lower limits in the desired zone.
G21: Defines the upper limits in the desired zone.

The format to program these functions is:

G20 K X...C +/- 5.5
G20 K X...C +/- 5.5

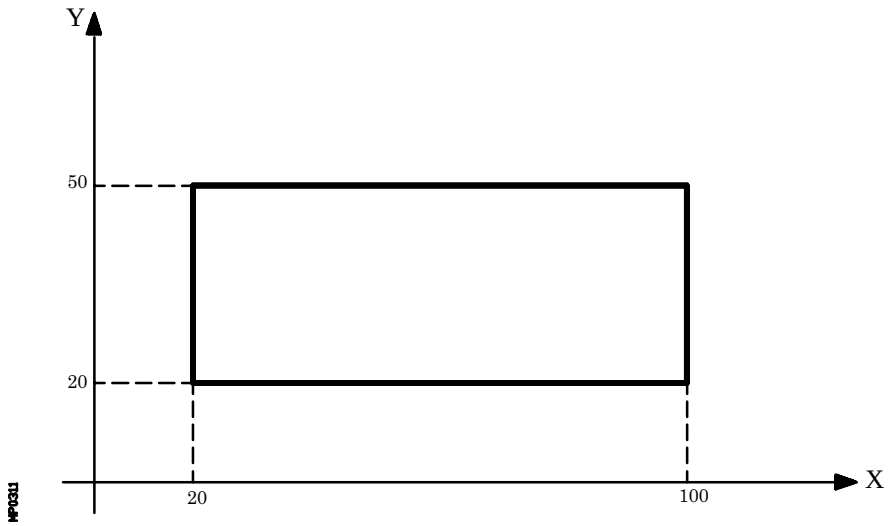
In which :

*** K** Indicates the work zone you wish to define (1, 2, 3 or 4)

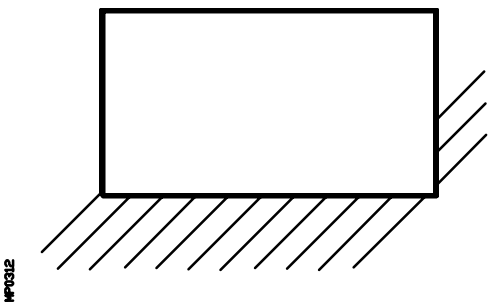
*** X...C** Indicates the coordinates (upper or lower) with which you wish to limit the axes. These coordinates will be programmed with reference to machine zero (home).

It is not necessary to program all the axes, so only defined axes will be limited.

Example:



G20 K1 X20 Y20
G21 K1 X100 Y50



4. REFERENCE SYSTEMS

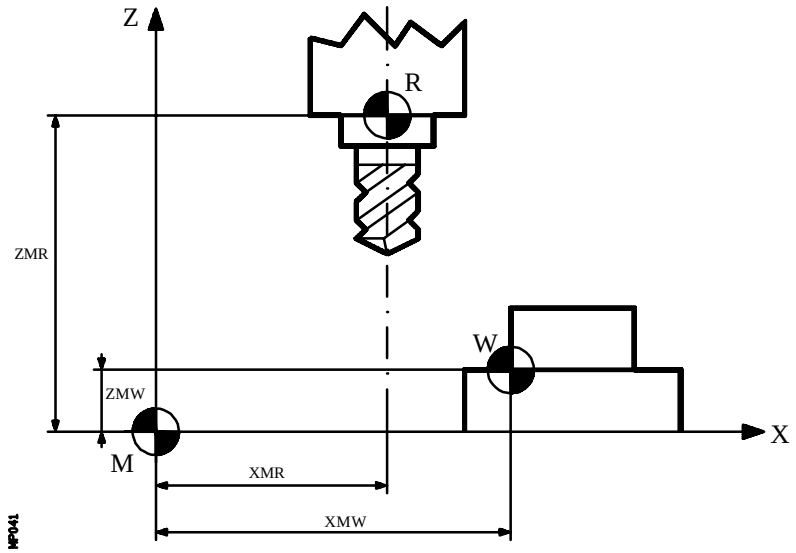
4.1 REFERENCE POINTS

A CNC machine needs the following origins and reference points defined :

* **Machine Reference Zero** or home. This is set by the manufacturer as the origin of the machine's coordinate system.

* **Part zero** or point of origin of the part. This is the point of origin which is set for programming the measurements of the part. It can be freely selected by the programmer, and its value with respect to machine zero can be set by the zero offset.

* **Machine Reference point.** This is a point on the machine established by the manufacturer around which the synchronization of the system is done. The control positions the axis on this point, instead of moving it as far as the Machine Reference Zero, taking, at this point, the reference coordinates which are defined via the axis machine parameter "REFVALUE".



M	Machine reference zero
W	Part zero
R	Machine reference point
XMW,YMW,ZMW, etc.	Coordinates of part zero
ZMR,YMR,ZMR, etc.	Coordinates of machine reference point ("REFVALUE")

4.2 MACHINE REFERENCE SEARCH (G74)

The CNC allows you to program the machine reference search in two ways :

* **MACHINE REFERENCE SEARCH OF ONE OR MORE AXES IN A PARTICULAR ORDER**

  G74 is programmed followed by the axes in which you want to carry out the reference search. For example : G74 X Z C Y

  The CNC begins the movement of all the selected axes which have a machine reference switch (machine axis parameter “DECINPUT”) and in the direction indicated by the axis machine parameter “REFDIREC”.

  This movement is carried out at the feedrate indicated by the axis machine parameter “REFEED1” for each axis until the home switch is hit.

  Next, the home search (marker pulse or home) will be carried out in the programmed order.

  This second movement will be carried out one axis at a time, at the feedrate indicated in the axis machine parameter “REFEED2” until the machine reference point is reached (i.e. the marker pulse is found).

* **MACHINE REFERENCE SEARCH USING THE ASSOCIATED SUBROUTINE.**

  The G74 function will be programmed alone in the block, and the CNC will automatically execute the subroutine whose number appears in the general machine parameter “REFPSUB”. In this subroutine it is possible to program the machine reference searches required, and also in the required order.

In a block in which G74 has been programmed, no other preparatory function may appear.

If the machine reference search is done in JOG mode, the part zero selected is lost. The coordinates of the reference point indicated in the machine axis parameter “REFVALUE” is displayed. In all other cases, the part zero selected is maintained, so the displayed coordinates refer to this part zero.

If the G74 command is executed in MDI, the display of coordinates depends on the mode in which it is executed : Jog, Execution, or Simulation.

4.3 PROGRAMMING WITH RESPECT TO MACHINE ZERO (G53)

Function G53 can be added to any block which has path control functions.

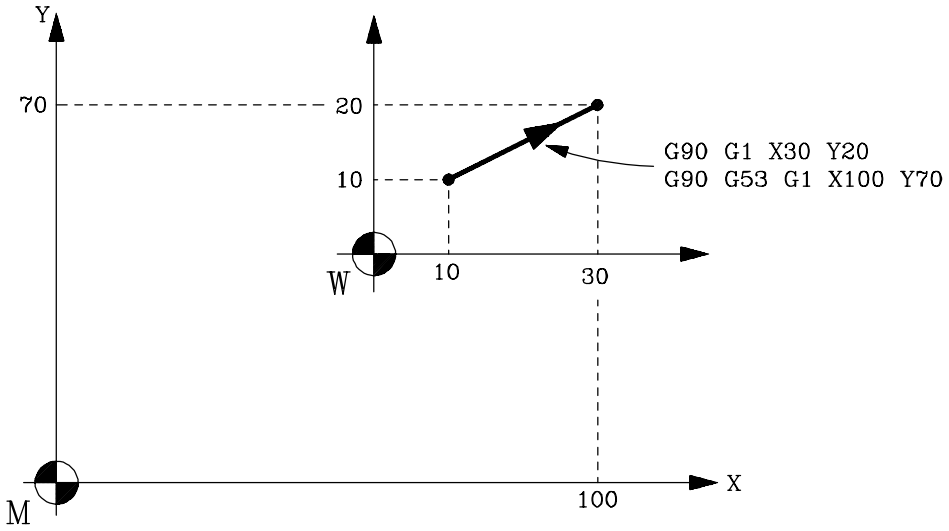
It is only used when the programming of block coordinates relating to machine zero is required. These coordinates should be expressed in millimeters or inches, depending on how the general machine parameter “INCHES” is defined.

By programming G53 alone (without motion information) the current active zero offset is canceled regardless of whether it was originated by a G54-G59 or a G92 preset. This G92 origin preset is described next. Once a Zero Offset has been selected, it will remain active until another one is selected or until a home search is carried out (G74). This Zero Offset will remain active even after powering the CNC off.

Function G53 is not modal, so it should be programmed every time you wish to indicate the coordinates referred to machine zero.

This function temporarily cancels radius and tool length compensation.

Example:

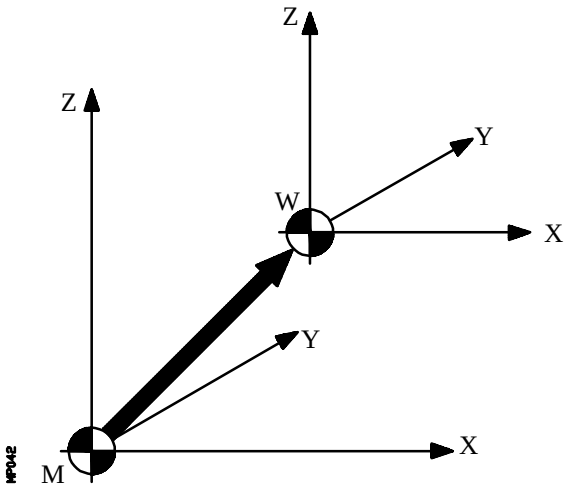


M Machine Reference Zero (home)
W Part Zero.

4.4 PRESETTING OF COORDINATES AND ZERO OFFSETS

The CNC allows you to carry out zero offsets with the aim of using coordinates related to the plane of the part, without having to modify the coordinates of the different points of the part at the time of programming.

The zero offset is defined as the distance between the part zero (point of origin of the part) and the machine zero (point of origin of the machine).



M Machine zero
W Part zero

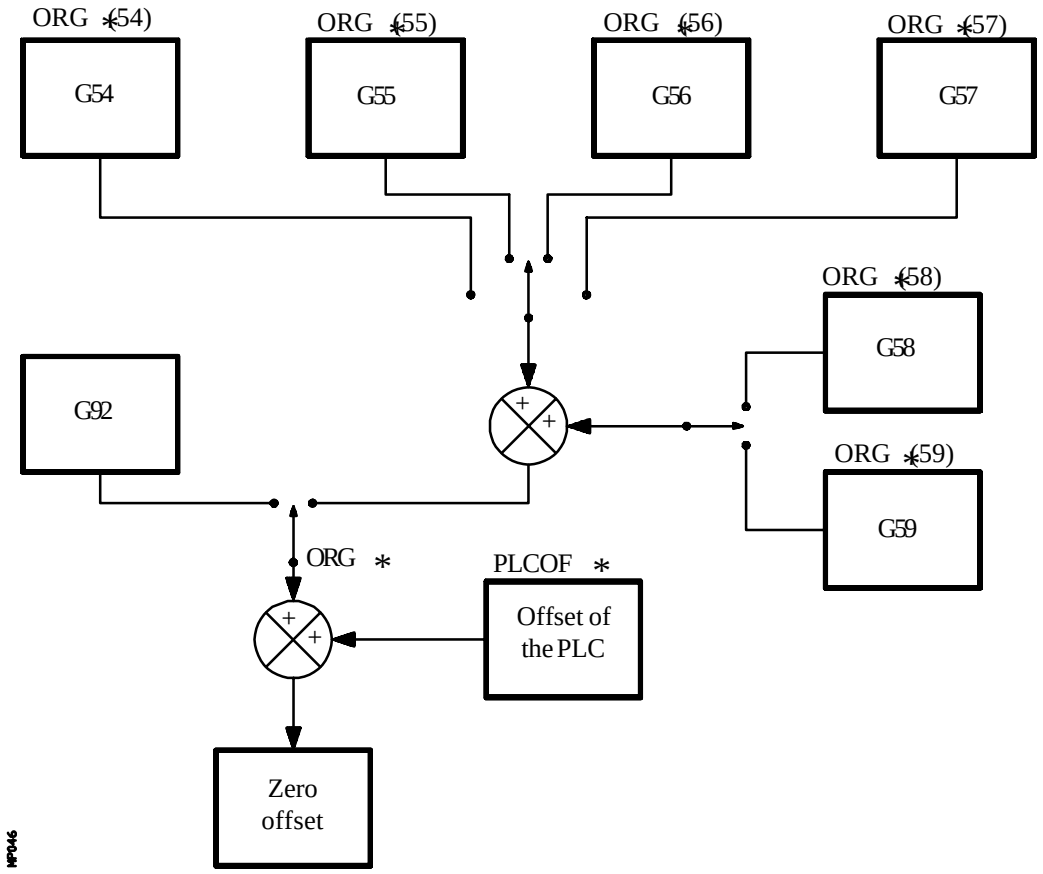
This zero offset can be carried out in one of two ways :

* Via Function G92 (coordinate preset). The CNC accepts the coordinates of the programmed axes after G92 as new axis values.

* Via the use of zero offsets (G54,G55,G56,G57,G58, G59). The CNC accepts as a new part zero the point located relative to machine zero at the distance indicated by the selected table(s).

Both functions are modal and incompatible, so if one is selected the other is disabled.

There is, moreover, another zero offset which is governed by the PLC. This offset is always added to the zero offset selected and is used (among other things) to correct deviations produced as a result of expansion, etc.


ORG *(54)
G54
ORG *(55)
G55
ORG *(56)
G56
ORG *(57)
G57
ORG *(58)
G58
G92
ORG *(59)
G59
ORG *
PLCOF *
Offset of the PLC
Zero offset
MP046

4.4.1 COORDINATE PRESET AND LIMITATION OF THE S VALUE (G92)

Via Function G92 one can select any value in the axes of the CNC, in addition to limiting the spindle speed.

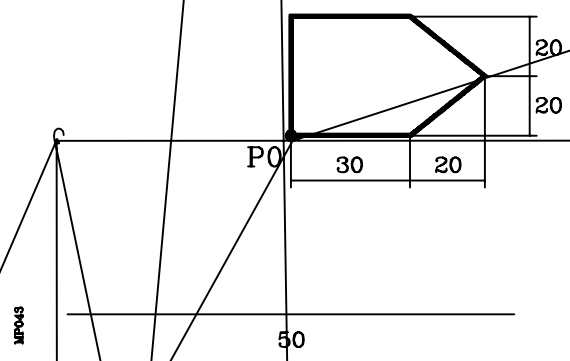
*** COORDINATE PRESET**

When carrying out a zero offset via Function G92, the CNC assumes the coordinates of the axes programmed after G92 as new axis values.

No other function can be programmed in the block where G92 is defined, the programming format being :

G92X...C +/- 5.5

Example :




```
G90  X50   Y40    ; Positioning in P0
G92  X0    Y0     ; Preset P0 as part zero
G91  X30           ; Programming according to part coordinates
     X20   Y20
     X-20  Y20
     X-30
           Y-40
```

*** LIMITATION OF SPINDLE SPEED**

When executing a "G92 S5.4" type block, the CNC limits the spindle speed from that instant on to the value set by S5.4.

If later on, a block is to be executed at a greater "S", the CNC will execute that block at the maximum "S" set with function G92S

Neither is it possible to exceed this maximum value from the keyboard on the front panel.

4.4.2 ZERO OFFSETS (G54..G59)

The CNC has a table of zero offsets, in which several zero offsets can be selected. The aim is to generate certain part zeros independently of the part zero active at the time.

Access to the table can be obtained from the front panel of the CNC (as explained in the Operating Manual), or via the program using high-level language commands.

There are two kinds of zero offsets :

Absolute zero offsets (G54,G55,G56 & G57), which must be referred to machine zero.

Additive zero offsets (G58,G59).

Functions G54, G55, G56, G57, G58 & G59 must be programmed alone in the block, and work in the following way:

When one of the G54, G55, G56, G57 functions is executed, the CNC applies the zero offset programmed with respect to machine zero, cancelling the possible active zero offsets.

If one of the additive offsets G58 or G59 is executed, the CNC adds its values to the absolute zero offset active at the time. Previously cancelling the additive offset which might be active.

You can see (in the following example) the zero offsets which are applied when the program is executed.

```
G54  Applies zero offsets G54 ------------------ >  G54
G58  Adds zero offsets G58 ---------------------- >  G54+G58
G59  Cancels G58 and adds G59 ------------------- >  G54+G59
G55  Cancels whatever and applies G55 ------ >  G55
```

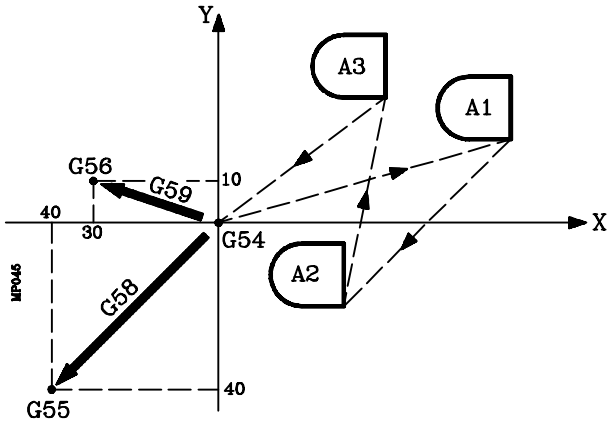
Once a Zero Offset has been selected, it will remain active until another one is selected or until a home search is carried out (G74) in JOG mode. This Zero Offset will remain active even after powering the CNC off.

This kind of zero offsets established by program is very useful for repeated machining operations at different machine positions.

Example :

The zero offset table is initialized with the following values:

G54:	X200	Y100			
G55:	X160	Y 60	G58:	X-40	Y-40
G56:	X170	Y110	G59:	X-30	Y 10



Using absolute zero offsets:

G54	; Applies G54 offset
Profile execution	; Executes profile A1
G55	; Applies G55 offset
Profile execution	; Executes profile A2
G56	; Applies G56 offset
Profile execution	; Executes profile A3

Using incremental zero offsets:

G54	; Applies G54 offset
Profile execution	; Executes profile A1
G58	; Applies offsets G54+G58
Profile execution	; Executes profile A2
G59	; Applies offsets G54+G59
Profile execution	; Executes profile A3

4.5 POLAR ORIGIN PRESET (G93)

Function G93 allows you to preset any point from the work plane as a new origin of polar coordinates.

This function must be programmed alone in the block, its format being :

G93 I+/-5.5 J+/-5.5

Parameters I & J respectively define the abscissa and ordinate axes, of the new origin of polar coordinates.

Example :

Assuming that the tool is at X0 Y0

```
G93       I35  J30  ; Preset P3 as polar origin
G90 G01   R25  Q0   ; Point P1, in a straight line (G01)
    G03        Q90  ; Point P2, in an arc (G03)
    G01   X0   Y0   ; Point P0, in a straight line (G01)
```

If G93 is only programmed in a block, the point where the machine is at that moment becomes the polar origin.

Warning:

The CNC does not modify the polar origin when defining a new part zero; but it modifies the values of the variables: "PORGF" y "PORGS".

If, while selecting the general machine parameter “PORGMOVE” a circular interpolation is programmed (G02 or G03), the CNC assumes the center of the arc as the new polar origin.

On power-up; or after executing M02, M30; or after an EMERGENCY or RESET; the CNC assumes the currently active part zero as polar origin.

When selecting a new work plane (G16, G17, G18, G19), the CNC assumes as polar origin the part zero of that plane.30P2P1P3P02535

5. *PROGRAMMING BY ISO CODE*

A programmed block in ISO language can consist of :

Preparatory functions **(G)**
Axis coordinates **(X...C)**
Feedrate **(F)**
Spindle speed **(S)**
Tool number **(T)**
Tool offset number **(D)**
Auxiliary functions **(M)**

This order should be maintained within each block, although it is not necessary for every block to contain the information.

The CNC allows you to program figures from 0.00001 to 99999.9999 with or without sign, working in millimeters (G71), called format +/-5.4, or either from 0.00001 to 3937.00787 with or without sign if the programming is done in inches (G70), called format +/-4.5.

Nevertheless, and in order to simplify explanations, we can say that the CNC admits Format +/-5.5, meaning that it admits +/-5.4 in millimeters and +/-4.5 in inches.

Any function with parameters can also be programmed in a block, apart from the number of the label or block. Thus, when the block is executed the CNC substitutes the arithmetic parameter for its value at that time.

5.1 PREPARATORY FUNCTIONS

Preparatory functions are programmed using the letter G followed by 2 digits.

They are always programmed at the beginning of the body of the block and are useful in determining the geometry and working condition of the CNC.

Table of G functions used in the CNC :

Function	M	D	V	Meaning	Section
G00	*	?	*	Rapid travel	6.1
G01	*	?	*	Linear interpolation	6.2
G02	*		*	Clockwise (helical) circular interpolation	6.3
G03	*		*	Counter-clockwise (helical) circular interpolation	6.3
G04				Dwell/block preparation stop	7.1, 7.2
G05	*	?	*	Round corner	7.3.1
G06			*	Absolute arc center coordinates	6.4
G07	*	?		Square corner	7.3.2
G08			*	Arc tangent to previous path	6.5
G09			*	Arc defined by three points	6.6
G10	*	*		Mirror image cancellation	7.5
G11	*		*	Mirror image on X axis	7.5
G12	*		*	Mirror image on Y axis	7.5
G13	*		*	Mirror image on Z axis	7.5
G14	*		*	Mirror image in the programmed directions	7.5
G15	*		*	Longitudinal axis selection	8.2
G16	*		*	Selection of main plane in two directions	3.2
G17	*	?	*	Main plane X-Y and longitudinal Z	3.2
G18	*	?	*	Main plane Z-X and longitudinal Y	3.2
G19	*		*	Main plane Y-Z and longitudinal X	3.2
G20				Definition of lower work zone limits	3.7.1
G21				Definition of upper work zone limits	3.7.1
G22			*	Activate/cancel work zones	3.7.2
G23			*	Activate tracing	16.3
G24			*	Activate digitizing	16.6
G25				Deactivate tracing/digitizing	16.5
G26			*	Tracing probe calibration	16.2
G27			*	Tracing contour definition	16.4
G28	*		*	Second spindle selection	5..3
G29	*	*		Main spindle selection	5..3
G28-G29			*	Axis toggle	7.9
G32	*		*	Feedrate as an inverted function of time.	6.14
G33	*		*	Threadcutting	6.12
G36			*	Automatic radius blend	6.10
G37			*	Tangential entry	6.8
G38			*	Tangential exit	6.9
G39			*	Automatic chamfer blend	6.11
G40	*	*		Cancellation of tool radius compensation	8.1
G41	*		*	Right-hand tool radius compensation	8.1
G42	*		*	Left-hand tool radius compensation	8.1
G43	*	?	*	Tool length compensation	8.2
G44	*	?	*	Cancellation of tool length compensation	8.2
G45	*		*	Tangential control	6.15
G47			*	Tool movement acoording tool coordinate system	17.2
G48	*		*	TCP transformation	17.3
G49	*		*	Incline plane definition	17.1
G50	*		*	Controlled corner rounding	7.3.3
G51	*		*	Look-Ahead	7.4
G52				Movement until making contact	6.13
G53				Program coordinates with respect to home	4.3

Function	M	D	V	Meaning	Section
G54	*		*	Absolute zero offset 1	4.4.2
G55	*		*	Absolute zero offset 2	4.4.2
G56	*		*	Absolute zero offset 3	4.4.2
G57	*		*	Absolute zero offset 4	4.4.2
G58	*		*	Additive zero offset 1	4.4.2
G59	*		*	Additive zero offset 2	4.4.2
G60			*	Straight line canned cycle	10.1
G61			*	Rectangular pattern canned cycle	10.2
G62			*	Grid pattern canned cycle	10.3
G63			*	Circular pattern canned cycle	10.4
G64			*	Arc pattern canned cycle	10.5
G65			*	Arc-chord pattern canned cycle	10.6
G66			*	Irregular pocket canned cycle	11.1
G67			*	Irregular pocket roughing	11.3
G68			*	Irregular pocket finishing	11.4
G69	*		*	Complex deep hole drilling	9.5.1
G70	*	?	*	Programming in inches	3.3
G71	*	?		programming in millimeters	3.3
G72	*		*	General and specific scaling factor	7.6
G73	*		*	Pattern rotation	7.7
G74			*	Machine reference search	4.2
G75			*	Probing until touching	12.1
G76			*	Probing while touching	12.1
G77	*		*	Slaved axis	7.8.1
G78	*	*		Slaved axis cancellation	7.8.2
G79				Canned cycle parameter modification	9.2.1
G80	*	*		Canned cycle cancellation	9.3
G81	*		*	Drilling cycle	9.5.2
G82	*		*	Drilling cycle with dwell	9.5.3
G83	*		*	Simple deep hole drilling	9.5.4
G84	*		*	Tapping cycle	9.5.5
G85	*		*	Reaming cycle	9.5.6
G86	*		*	Boring cycle with withdrawal in G00	9.5.7
G87	*		*	Rectangular pocket milling cycle	9.5.8
G88	*		*	Circular pocket milling cycle	9.5.9
G89	*		*	Boring cycle with withdrawal in G01	9.5.10
G90	*	?		Programming in absolute	3.4
G91	*	?	*	Programming in incremental	3.4
G92				Coordinate preset/spindle speed limit	4.4.1
G93				Polar origin preset	4.5
G94	*	?		Feedrate in millimeters(inches) per minute	5.2.1
G95	*	?	*	Feedrate in millimeters(inches) per revolution	5.2.2
G96	*		*	Constant cutting point speed	5.4.1
G97	*	*		Constant tool center speed	5.4.2
G98	*	*		Withdrawal to the starting plane	9.5
G99	*		*	Withdrawal to the reference plane	9.5

M means modal, i.e. the G function, once programmed, remains active while another incompatible G function is not programmed.

D means BY DEFAULT, i.e. they will be assumed by the CNC when it is powered on, after executing M02, M30 or after EMERGENCY or RESET.

In those cases indicated by **?** , it should be understood that the DEFAULT of these G functions depends on the setting of the general machine parameters of the CNC.

V means that the G code is displayed next to the current machining conditions in the execution and simulation modes.

5.2 FEEDRATE FUNCTIONS (G94,G95)

The CNC allows programming the feedrate of the axes in mm/minute and mm/revolution when working in millimeters, or in inches/minute and inches/revolution when working in inches.

5.2.1 FEEDRATE IN MM/MIN OR INCHES/MIN (G94)

From the moment the code G94 is programmed, the control takes that the feedrates programmed through **F5.5** are in mm/min or inches/mm.

If the movement corresponds to a rotary axis, the CNC interprets the feedrate as being programmed in degrees/min.

If an interpolation is made between a rotary and a linear axis, the programmed feedrate is taken in mm/min or inches/min, and the movement of the rotary axis (programmed in degrees) will be considered programmed in millimeters or inches.

The relationship between the feedrate of the axis component and the programmed feedrate "F" is the same as that between the movement of the axis and the resulting programmed movement.

$$\text{Feedrate component} = \frac{\text{Feedrate F x Movement of axis}}{\text{Resulting programmed movement}}$$

Example :

On a machine which has linear X and Y axes and rotary C axis, all located at point X0 Y0 C0, the following movement is programmed :

G1 G90 X100 Y20 C270 F10000

You get:

$$\frac{F\ \triangle x}{\sqrt{(\triangle x)^2+(\triangle y)^2+(\triangle c)^2}} \quad \frac{10000 \times 100}{\sqrt{100^2+20^2+270^2}} = 3464.7946$$

$$\frac{F\ \triangle y}{\sqrt{(\triangle x)^2+(\triangle y)^2+(\triangle c)^2}} \quad \frac{10000 \times 20}{\sqrt{100^2+20^2+270^2}} = 692.9589$$

$$\frac{F\ \triangle c}{\sqrt{(\triangle x)^2+(\triangle y)^2+(\triangle c)^2}} \quad \frac{10000 \times 270}{\sqrt{100^2+20^2+270^2}} = 9354.9455$$

Function G94 is modal i.e. once programmed it stays active until G95 is programmed.

On power-up, after executing M02, M30 or following EMERGENCY or RESET, the CNC assumes function G94 or G95 according to how the general machine parameter "IFEED" is set.

5.2.2 FEEDRATE IN MM/REV.OR INCHES/REV (G95)

From the moment when the code G95 is programmed, the control assumes that the feedrates programmed through **F5.5** are in mm/rev or inches/mm.

This function does not affect the rapid moves (G00) which will be made in mm/min or inch/min. By the same token, it will not be applied to moves made in the JOG mode, during tool inspection, etc.

Function G95 is modal i.e. once programmed it stays active until G94 is programmed.

On power-up, after executing M02, M30 or following EMERGENCY or RESET, the CNC assumes function G94 or G95 according to the general machine parameter "IFEED".

5.3 SPINDLE SELECTION (G28, G29)

This CNC can govern two spindles: the main one and the second one. Both can be running at the same time but it can only control one at the time.

This selection is done by functions: G28 and G29.

G28: Selects the Second Spindle
G29: Selects the Main Spindle.

Once the desired spindle has been selected, it can be acted upon from the keyboard or by means of the following functions:

M3, M4, M5, M19
S****
G33, G94, G95, G96, G97

Both spindles can work in open and closed loop.

Functions G28 and G29 are modal and incompatible with each other.

Function G28 and G29 must be programmed alone in the block. No more information can be programmed in that block.

On power-up, after executing and M02, M30 or after an EMERGENCY or RESET, the CNC assumes function G29 (selects the main spindle).

Operating example for when 2 spindles are used:

On power-up, the CNC assumes function G29 selecting the main spindle.
All the actions upon the keys or functions associated with the spindle will be applied on to the main spindle.
Example: S1000 M3 Main spindle clockwise at 1000 rpm

To select the second spindle, execute function G28.
From this moment on, all the actions upon the keys or functions associated with the spindle will be applied on to the second spindle.
The main spindle keeps turning (in its previous status).
Example: S1500 M4 Second spindle counter-clockwise at 1500 rpm.
The main spindle keeps turning clockwise at 1000 rpm

To select the main spindle again, execute function G29.

From this moment on, all the actions upon the keys or functions associated with the spindle will be applied on to the main spindle.
The second spindle keeps turning (in its previous status).
Example: S2000 The main spindle keeps turning clockwise but now at 2000 rpm.
The second spindle keeps turning counter-clockwise at 1500 rpm.

5.4 CONSTANT SPEED FUNCTIONS (G96,G97)

The CNC, through functions G96 and G97, allows you to maintain constant speed at tool center, or maintain constant speed of the cutting point of the tool.

5.4.1 CONSTANT SURFACE SPEED (G96)

When G96 is programmed the CNC takes the **F5.5** feedrate as corresponding to the cutting point of the tool on the part.

By using this function, the finished surface is uniform in curved sections.

In this manner (working in function G96) the speed of the center of the tool in the inside or outside curved sections will change in order to keep the cutting point constant.

Function G96 is modal i.e. once programmed, it is active until G97 is programmed.

On power-up, after executing M02, M30 or following EMERGENCY or RESET, the CNC assumes function G97.

5.4.2 CONSTANT TOOL-CENTER SPEED (G97)

When G97 is programmed the CNC takes the programmed **F5.5** feedrate as corresponding to the feedrate of the center of the tool.

In this manner (working in function G97) the speed of the cutting point on the inside or outside curved sections is reduced, keeping the speed of the center of the tool constant.

Function G97 is modal i.e. once programmed it is active until G96 is programmed.

On power-up, after executing M02, M30 or following EMERGENCY or RESET, the CNC assumes function G97.

5.5 COMPLEMENTARY FUNCTIONS

The CNC is equipped with the following complementary functions :

Feedrate **F**
Spindle speed **S**
Tool number **T**
Tool offset number **D**
Miscellaneous function **M**

This order should be maintained in each block, although it is not necessary for each block to hold all the information.

5.5.1 FEEDRATE F

The machining feedrate can be selected from the program. It remains active until another feedrate is programmed.

It is represented by the letter **F**. Depending on whether it is working in G94 or G95, it is programmed in mm/minute (inches/minute) or in mm/revolution (inches/revolution).

It's programming format is 5.5 in mm. and 4.5 in inches.

The maximum operating feedrate of the machine, limited on each axis by the axis machine parameter "MAXFEED", may be programmed via code F0, or by giving F the corresponding value.

The programmed feedrate F is effective working in linear (G01) or circular (G02, G03) interpolation. If function F is not programmed, the CNC assumes the feedrate to be F0. When working in rapid travel (G00), the machine will move at the rapid feedrate indicated by the axis machine parameter "G00FEED", apart from the F programmed.

The programmed feedrate F may be varied between 0% and 255% via the PLC, or by DNC, or between 0% and 120% via the switch located on the Operator Panel of the CNC.

The CNC, however, is equipped with the general machine parameter "MAXFOVR" to limit maximum feedrate variation.

If you are working in rapid travel (G00), rapid feedrate will be fixed at 100%, alternatively it can be varied between 0% and 100%, depending on how the machine parameter "RAPIDOVR" is set.

When functions G33 (electronic threading) or G84 (tapping canned cycle) are executed the feedrate cannot be modified. It functions at 100% of programmed F.

5.5.2 SPINDLE SPEED AND SPINDLE ORIENTATION (S)

Code S has two meanings :

a) TURNING SPEED OF THE SPINDLE

The turning speed of the spindle is programmed directly in rpm via code S5.4.

The maximum value is limited by spindle machine parameters “MAXGEAR1”, MAXGEAR2, MAXGEAR 3 and MAXGEAR4", in each case depending on the spindle range selected.

It is also possible to limit this maximum value from the program by using function G92 S5.4.

The programmed turning speed S may be varied from the PLC, DNC, or by the SPINDLE keys “+” and “-” on the Operator Panel of the CNC.

This speed variation is made between the maximum and minimum values established by spindle machine parameters “MINSOVR and MAXSOVR”.

The incremental pitch associated with the SPINDLE keys “+” and “-” on the CNC Operator Panel in order to vary the programmed S value is fixed by the spindle machine parameter “SOVRSTEP”.

When functions G33 (threading) or G84 (tapping cycle) are executed the speed cannot be modified. It functions at 100% of programmed S.

b) SPINDLE ORIENTATION

If S±5.5 is programmed after M19, code S±5.5 indicates the spindle orientation position in degrees starting from the machine reference pulse from the encoder.

To carry out this function you need a rotary encoder coupled to the machine spindle.

If you do not have a reference switch, the spindle moves at the turning speed indicated by the spindle machine parameter “REFEED1” until the spindle is located at the point defined via S±5.5.

If you have a ref. switch, the spindle moves at the turning speed indicated by spindle machine parameter “REFEED1” until it reaches the switch, and then at the one indicated by spindle machine parameters “REFEED2” until the spindle is at the point defined via S±5.5.

The “REFEED1” movement until the reference switch is reached is always done provided M19 is programmed after the spindle has operated in open loop (M3, M4, M5). This movement is not made between consecutive M19s.

5.5.3 TOOL NUMBER (T)

The CNC enables you to select the tool or tools required for each machining operation via function **T4**.

There is a tool magazine table whose number of components is established by "NPOCKET" (general machine parameter), specifying the following for each component:

* The contents of the box, indicating tool number or if the box is empty or cancelled.

* The size of the tool. N if it is a normal tool and S if it is special.

* The status of the tool. A if it is available, E if it is worn out (life expired) and R if it has been rejected.

  It also has a tool table. The number of components in this table is established by "NTOOL" (general machine parameter), specifying the following for each component:

* The offset associated with each tool:

family code 0<n<200 —> normal tool
family code 200<n<255—> special tool

* Nominal tool life calculated for this tool, defined in terms of machining minutes or depending on the number of operations to be carried out. This is selected by the general machine parameter "TOOLMONI".

* Real working life of the tool, defined in machining minutes or depending on the number of operations carried out. This is selected by the general machine parameter "TOOLMONI".

* The size of the tool. N if it is a normal tool and S if it is special.

* The status of the tool. A if it is available, E if it is worn out (life expired) and R if it has been rejected by the PLC.

5.5.4 TOOL OFFSET NUMBER (D)

By using function **D4**, the CNC enables the application of the required offset to the selected tool. To do this you have to program T4 D4, selecting the required tool and tool offset. If you only program function T4, the CNC will take the tool offset indicated by the selected tool in the tool table.

The CNC has a tool offset table with “NTOFFSET” (general machine parameter) components. The following is specified for each tool offset:

* The nominal radius of the tool in the measuring units indicated by the general parameter “INCHES”, the format being R±5.5.

* The nominal length of the tool in the measuring units indicated by the general parameter “INCHES”, the format being L±5.5.

* Tool radius wear in the measuring units indicated by the general parameter “INCHES”, the format being I±5.5. The CNC adds this value to the nominal radius (R) to calculate the real radius (R+I).

* Tool length wear in the measuring units indicated by the general parameter “INCHES”, the format being K±5.5. The CNC adds this value to the nominal length (L) to calculate the real length (L+K).

When tool radius compensation is required (G41 or G42) the CNC applies as a compensation value the sum of the R+I values of the selected tool offset.

When compensation of tool length is required (G43) the CNC applies as a compensation value the sum of the L+K values of the selected tool offset.

If no tool offset has been defined, the CNC applies tool offset D0, with R=0, L=0, I=0 and K=0.

5.5.5 MISCELLANEOUS FUNCTION (M)

The miscellaneous functions are programmed by means of the **M4** Code, it being possible to program up to 7 functions in the same block.

When more than one function has been programmed in one block, the CNC executes these correlatively to the order in which they have been programmed.

The CNC is provided with an M functions table with "NMISCFUN" (general machine parameter) components, specifying for each element:

* The number (0-9999) of the defined miscellaneous M function.

* The number of the subroutine which is required to associate to this miscellaneous function.

* An indicator which determines if the M function is executed before or after the movement block in which it is programmed.

* An indicator which determines if the execution of the M function interrupts block preparation or not.

* An indicator which determines if the M function is executed or not, after the execution of the associated subroutine.

* An indicator which determines if the CNC must wait for the signal AUX END or not (Executed M signal, coming from the PLC), to continue the execution of the program.

If, when executing the M miscellaneous function, this is not defined in the M functions table, the programmed function will be executed at the beginning of the block and the CNC will wait for the AUX END to continue the execution of the program.

Some of the miscellaneous functions are assigned an internal meaning in the CNC.

If, while executing the associated subroutine of an "M" miscellaneous function, there is a block containing the same "M", this will be executed but not the associated subroutine.

Warning:

All the miscellaneous "M" functions which have an associated subroutine must be programmed alone in a block.

5.5.5.1 M00. PROGRAM STOP

When the CNC reads code M00 in a block, it interrupts the program. To start up again, press CYCLE START.

We recommend that you set this function in the table of M functions, in such a way that it is executed at the end of the block in which it is programmed.

5.5.5.2 M01. CONDITIONAL PROGRAM STOP

This is identical to M00, except that the CNC only takes notice of it if the signal M01 STOP from the PLC is active (high logic level).

5.5.5.3 M02. END OF PROGRAM

This code indicates the end of program and carries out a “General Reset” function of the CNC (returning it to original state). It also carries out the M05 function.

It is recommended to set this function in the table of M functions, in such a way that it is executed at the end of the block in which it is programmed.

5.5.5.4 M30. END OF PROGRAM WITH RETURN TO FIRST BLOCK

Identical to M02 except that the CNC returns to the first block of the program.

5.5.5.5 M03. CLOCKWISE SPINDLE ROTATION

This code represents clockwise spindle start. As explained in the corresponding section, the CNC automatically executes this code in the machining canned cycles.

It is recommended to set this function in the table of M functions, so that it is executed at the beginning of the block in which it is programmed.

5.5.5.6 M04. COUNTER-CLOCKWISE SPINDLE ROTATION

This code represents counter-clockwise spindle start. We recommend that you set this function in the table of M functions, so that it is executed at the beginning of the block in which it is programmed.

5.5.5.7 M05. SPINDLE STOP

It is recommended to set this function in the table of M functions, so that it is executed at the end of the block in which it is programmed.

5.5.5.8 M06. TOOL CHANGE

If the general machine parameter “TOFFM06” (indicating that it is a machining center) is active, the CNC sends instructions to the tool changer and updates the table corresponding to the tool magazine.

It is recommended to set this function in the table of M functions, so that the subroutine corresponding to the tool changer installed in the machine is executed.

5.5.5.9 M19. SPINDLE ORIENTATION

With this CNC it is possible to work with the spindle in open loop (M3, M4) and with the spindle in closed loop (M19).

In order to work in closed loop, it is necessary to have a rotary encoder installed on the spindle of the machine.

To switch from open loop to closed loop, execute function M19 or M19 S±5.5. The CNC will act as follows:

* If the spindle does not have a home switch, the CNC changes the spindle speed until it reaches the one set by spindle machine parameter "REFEED2; finds the marker pulse (home) and, then, orients the spindle to the position defined by S±5.5.

* If the spindle has a home switch, the CNC modifies the spindle speed until it reaches the one set by spindle machine parameter "REFEED1". Then, it carries out the search for the home switch at this speed. Next, it looks for the marker pulse (home) at the speed set by spindle machine parameter "REFEED2" and, finally, it orients the spindle to the position defined by S±5.5.

If only M19 is executed, the spindle is oriented to position "S0" after having "found" the home switch.

To, now, orient the spindle to another position, program M19 S±5.5, the CNC will not perform the home search since it is already in closed loop and it will orient the spindle to the indicated position. (S±5.5).

The S±5.5 code indicates the spindle orient position, in degrees, from the encoder's marker pulse position (S0).

The sign indicates the counting direction and the 5.5 value is always considered to be absolute coordinates regardless of the type of units currently selected.

Example:

S1000 M3	Spindle in open loop
M19 S100	The spindle switches to closed loop. Home search and positioning (orientation) at 100º
M19 S-30	The spindle orients to -30º, passing through 0º.
M19 S400	The spindle turns a whole revolution and positions at 40º.

5.5.5.10 M41, M42, M43, M44. SPINDLE SPEED RANGE CHANGE

The CNC offers 4 spindle speed ranges M41, M42, M43 and M44 with maximum speed limits set by the spindle machine parameters “MAXGEAR1”, MAXGEAR2", “MAXGEAR3” and “MAXGEAR4”.

If machine parameter “AUTOGEAR” is set so the CNC executes the range change automatically, M41 thru M44 will be sent out automatically by the CNC without having to be programmed.

If this machine parameter is set for non-automatic gear change, M41 thru M44 will have to be programmed every time a gear change is required. Bear in mind that the maximum voltage value assigned to machine parameter “MAXVOLT” corresponds to the maximum speed indicated for each one of the speed ranges (machine parameters “MAXGEAR1” thru “MAXGEAR4”).

5.5.5.11 M45 AUXILIARY SPINDLE / LIVE TOOL

In order to use this miscellaneous function, it is necessary to set one of the axes of the machine as auxiliary spindle or live tool (general machine parameter P0 thru P7).

To use the auxiliary spindle or live tool, execute the command: **M45 S±5.5** where **S** indicates the turning speed in rpm and the sign indicates the turning direction.

The CNC will output the analog voltage corresponding to the selected speed according to the value assigned to the machine parameter "**MAXSPEED**" for the auxiliary spindle.

To stop the auxiliary spindle, program **M45** or **M45 S0**.

Whenever the auxiliary spindle or live tool is active, the CNC will let the PLC know by activating the general logic output "**DM45**" (M5548).

Also, it is possible to set the machine parameter for the auxiliary spindle "**SPDLOVR**" so the Override keys of the front panel can modify the currently active turning speed of the auxiliary spindle.

6. *PATH CONTROL*

The CNC allows you to program movements on one axis only or several at the same time.

Only those axes which intervene in the required movement are programmed. The programming order of the axes is as follows :

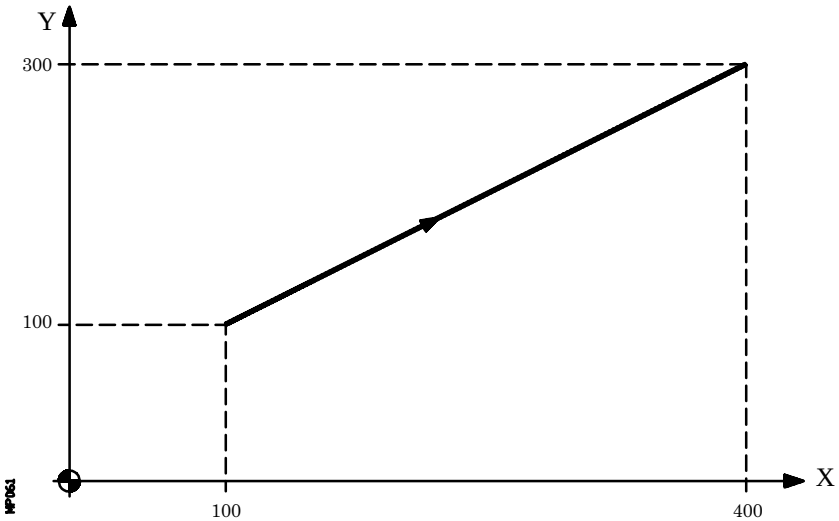
X, Y, Z, U, V, W, A, B, C

6.1 RAPID TRAVEL (G00)

The movements programmed after G00 are executed at the rapid feedrate indicated in the axis machine parameter "G00FEED".

Independently of the number of axis which move, the resulting path is always a straight line between the starting point and the final point.

Example:



X100 Y100 ; Starting point

G00 G90 X400 Y300 ; Programmed path

It is possible, via the general machine parameter "RAPIDOVR", to establish if the feedrate override % switch (when working in G00) operates from 0% to 100%, or whether it stays constant at 100%.

When G00 is programmed, the last "F" programmed is not cancelled i.e. when G01, G02 or G03 are programmed again "F" is recovered.

G00 is modal and incompatible with G01, G02, G03, G33 and G75. Function G00 can be programmed as G or G0.

On power-up, after executing M02, M30 or after EMERGENCY or RESET, the CNC assumes code G00 or G01, depending on how general machine parameter "IMOVE" has been set.

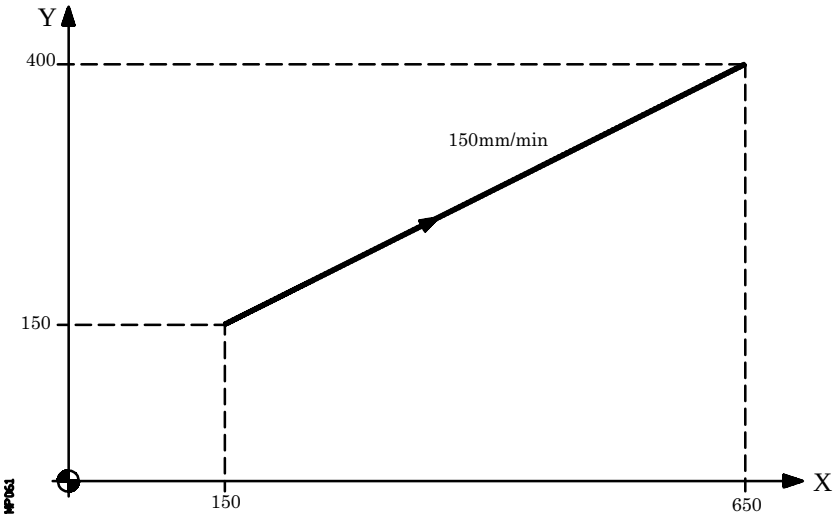
6.2 LINEAR INTERPOLATION (G01)

The movements programmed after G01 are executed according to a straight line and at the programmed feedrate "F".

When two or three axes move simultaneously the resulting path is a straight line between the starting point and the final point.

The machine moves according to this path to the programmed feedrate "F". The CNC calculates the feedrates of each axis so that the resulting path is the "F" value programmed.

Example:



G01 G90 X650 Y400 F150

The programmed feedrate "F" may vary between 0% and 120% via the switch located on the Control Panel of the CNC, or by selecting between 0% and 255% from the PLC, or via the DNC or the program.

Nevertheless, the CNC has general machine parameter "MAXFOVR" to limit maximum variation of the feedrate.

With this CNC, it is possible to program a positioning-only axis in a linear interpolation block. The CNC will calculate the feedrate for this positioning-only axis so it reaches the target coordinate at the same time as the interpolating axes.

Function G01 is modal and incompatible with G00, G02, G03, G33 and G75. Function G01 can be programmed as G1.

On power-up, after executing M02, M30 or after EMERGENCY or RESET, the CNC assumes code G00 or G01, depending on how general machine parameter "IMOVE" has been set.

6.3 CIRCULAR INTERPOLATION (G02. G03)

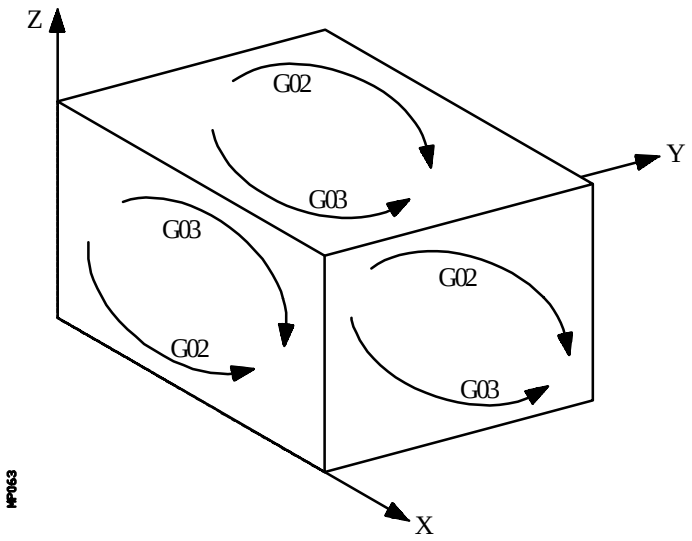
There are two ways of carrying out circular interpolation:

G02: Clockwise circular interpolation

G03: Counter-clockwise circular interpolation

Movements programmed after G02 and G03 are executed in the form of a circular path and at the programmed feedrate “F”.

Clockwise (G02) and counterclockwise (G03) definitions are established according to the system of coordinates shown below:



This system of coordinates refers to the movement of the tool on the part.

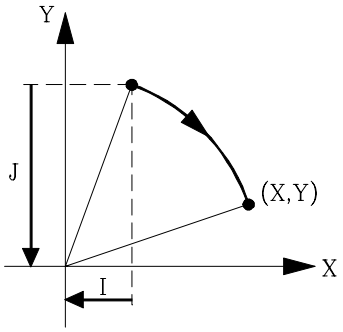
Circular interpolation can only be executed on a plane. The form of definition of circular interpolation is as follows :

a) CARTESIAN COORDINATES

The coordinates of the endpoint of the arc and the position of the center with respect to the starting point are defined according to the axes of the work plane.

The center coordinates (which should always be programmed even if they have "0" value) are defined by the letters I, J, or K, each one of these being associated to the axes as follows:

Axes X,U,A --> I
Axes Y,V,B —> J
Axes Z,W,C —> K



Programming format:

Plane XY: G02(G03) X±5.5 Y±5.5 I±5.5 J±5.5
Plane ZX: G02(G03) X±5.5 Z±5.5 I±5.5 K±5.5
Plane YZ: G02(G03) Y±5.5 Z±5.5 J±5.5 K±5.5

The programming order of the axes is always maintained regardless of the plane selected,, as are the respective center coordinates.

Plane AY: G02(G03) Y±5.5 A±5.5 J±5.5 I±5.5
Plane XU: G02(G03) X±5.5 U±5.5 I±5.5 I±5.5

b) POLAR COORDINATES

It is necessary to define the angle to be travelled **Q** and the distance from the starting point to the center (optional), according to the axes of the work plane.

The center coordinates are defined by the letters I, J, or K, each one of these being associated to the axes as follows:

Axes X,U,A —>I
Axes Y,V,B —> J
Axes Z,W,C —> K

If the center of the arc is not defined, the CNC will assume it that this coincides with the current polar origin.

Programming format:

Plane XY: G02(G03) Q±5.5 I±5.5 J±5.5
Plane ZX: G02(G03) Q±5.5 I±5.5 K±5.5
Plane YZ: G02(G03) Q±5.5 J±5.5 K±5.5

c) CARTESIAN COORDINATES WITH RADIUS PROGRAMMING

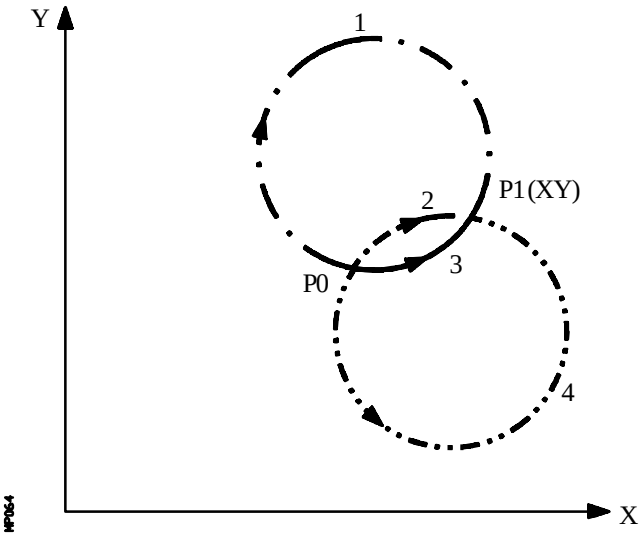
The coordinates of the endpoint of the arc and radius **R** are defined.

Programming format:

Plane XY: G02(G03) X±5.5 Y±5.5 R±5.5
Plane ZX: G02(G03) X±5.5 Z±5.5 R±5.5
Plane YZ: G02(G03) Y±5.5 Y±5.5 R±5.5

If a complete circle is programmed, with radius programming, the CNC will show the corresponding error, as infinite solutions exist.

If an arc is less than 180º, the radius is programmed with a plus sign, and a minus sign if it is more than 180º.

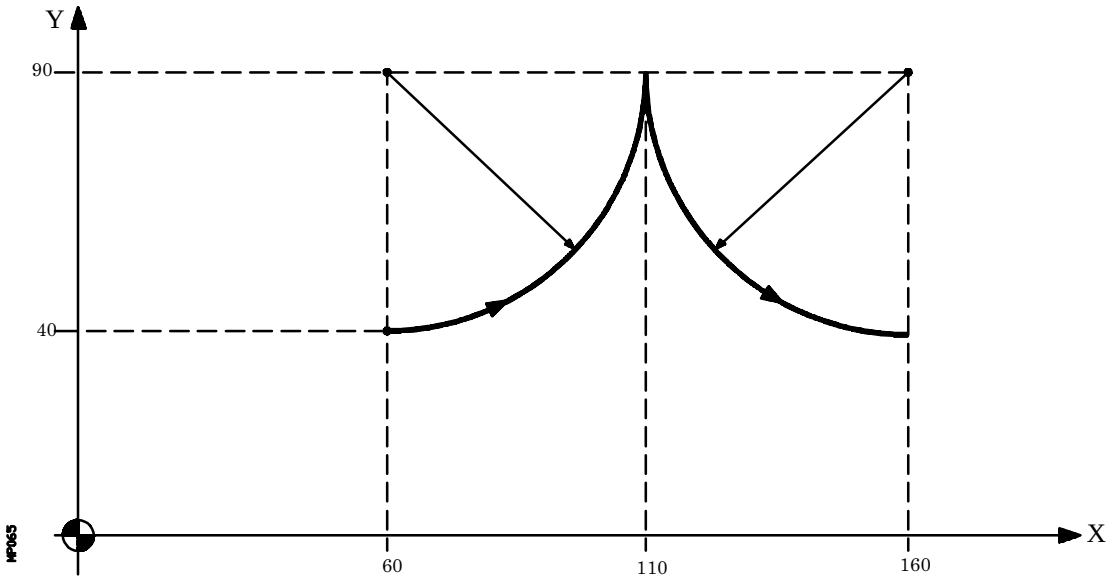


If P0 is the starting point and P1 the endpoint, there are 4 arcs which have the same value passing through both points.

Depending on the circular interpolation G02 or G03, and on the radius sign, the relevant arc is defined. Thus the programming format of the sample arcs is as follows:

Arc 1 G02 X.. Y.. R -..
Arc 2 G02 X.. Y.. R +..
Arc 3 G03 X.. Y.. R +..
Arc 4 G03 X.. Y.. R -..

Programming example:



Various programming modes are analyzed below, point X60 Y40 being the starting point.

Cartesian coordinates:

G90 G17 **G03 X110 Y90 I0 J50**
X160 Y40 I50 J0

Polar coordinates:

G90 G17 **G03 Q0 I0 J50**
Q-90 I50 J0

or:

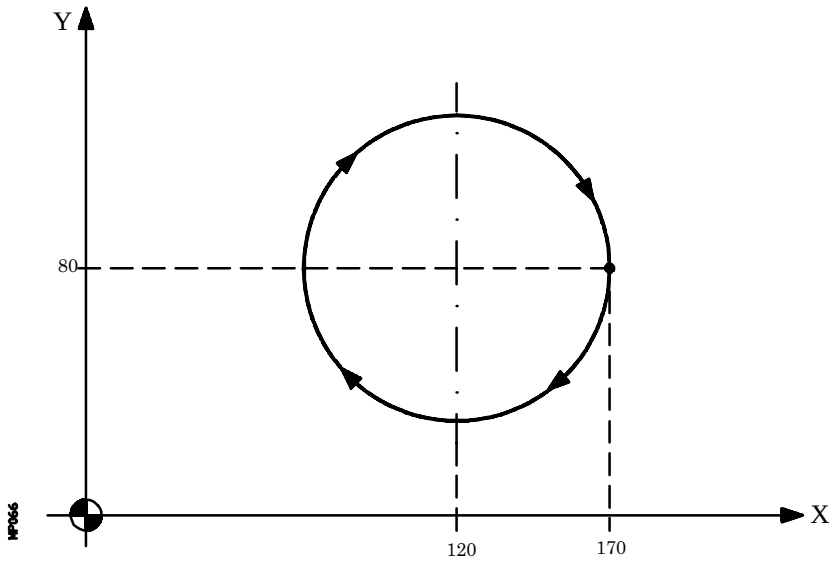
G93 I60 J90 ; defines polar center
G03 Q0
G93 I160 J90 ; defines new polar center
Q-90

Cartesian coordinates with radius programming:

G90 G17 **G03 X110 Y90 R50**
X160 Y40 R50

Example:

Programming of a (complete) circle in just one block:



Various programming modes analyzed below, point X170 Y80 being the starting Point.

Cartesian coordinates:

G90 G17 **G02 X170 Y80 I-50 J0**

or:

G90 G17 **G02 I-50 J0**

Polar coordinates:

G90 G17 **G02 Q360 I-50 J0**

or:

G93 I120 J80 ; defines polar center
G02 Q360

Cartesian coordinates with radius programming:

A complete circle cannot be programmed as there is an infinite range of solutions.

The CNC calculates, depending on the programmed arc, the radii of the starting point and endpoint. Although in theory both points should be exactly the same, the CNC enables you to select with the general machine parameter "CIRINERR", the maximum difference permissible between both radii, If this value is exceeded, the CNC displays the corresponding error.

The programmed feedrate "F" can be varied between 0% and 120% by using the switch located on the Operator Panel of the CNC, or by selecting it between 0% and 255% from the PLC, via the DNC or from the program.

The CNC, however, has general machine parameter "MAXFOVR" to limit the maximum variation of the feedrate.

If the general machine parameter "PORGMOVE" has been selected and a circular interpolation (G02 or G03) is programmed, the CNC assumes the center of the arc to be a new polar origin.

Functions G02 and G03 are modal and incompatible both among themselves and with G00, G01, and G33. Functions G02 and G03 can be programmed as G2 and G3.

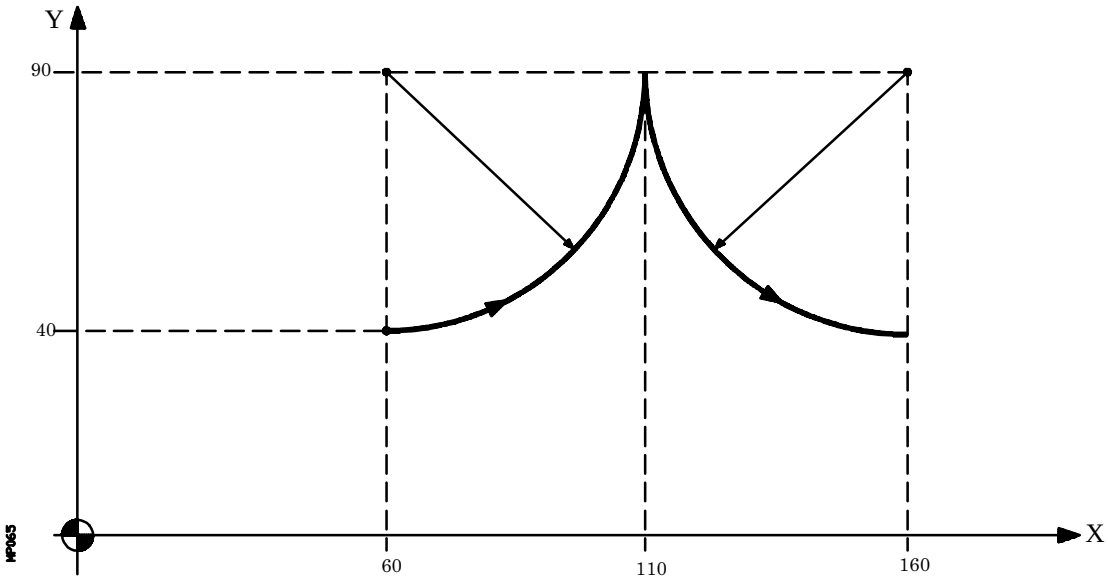
On power-up, after executing M02, M30 or after EMERGENCY or RESET, the CNC assumes code G00 or G01, depending on how general machine parameter "IMOVE" has been set.

6.4 CIRCULAR INTERPOLATION BY PROGRAMMING THE CENTER OF THE ARC IN ABSOLUTE COORDINATES (G06)

By adding function **G06** to a circular interpolation block you can program the coordinates of the center of the arc (I,J, or K) in absolute coordinates i.e. with respect to the zero origin and not to the beginning of the arc.

Function G06 is not modal, so it should be programmed any time the coordinates of the center of the arc are required in absolute coordinates. G06 can be programmed as G6.

Example:



Various programming modes are analyzed below, point X60 Y40 being the starting point.

Cartesian coordinates:

G90 G17 **G06** G03 X110 Y90 **I60 J90**
G06 X160 Y40 **I160 J90**

Polar coordinates:

G90 G17 **G06** G03 Q0 **I60 J90**
G06 Q-90 **I160 J90**

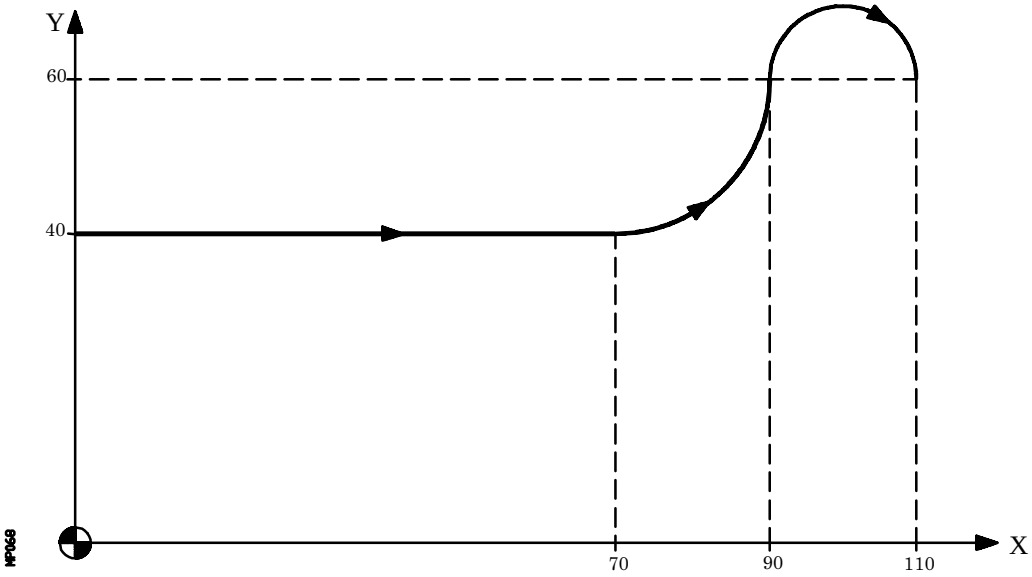
6.5 ARC TANGENT TO THE PREVIOUS PATH (G08)

Via function **G08** you can program an arc tangential to the previous path without having to program the coordinates (I.J &K) of the center.

Only the coordinates of the endpoint of the arc are defined, either in polar coordinates or in Cartesian coordinates according to the axes of the work plane.

Example:

Supposing that the starting point is X0 Y40, you wish to program a straight line, then an arc tangential to the line and finally an arc tangential to the previous one.




```
G90  G01  X70
     G08  X90  Y60   ; arc tangential to previous path
     G08  X110 Y60   ; arc tangential to previous path
```

Function G08 is not modal, so it should always be programmed if you wish to execute an arc tangential to the previous path. Function G08 can be programmed as G8.

Function G08 enables the previous path to be a straight line or an arc and does not alter its history. The same function G01, G02 or G03 stays active after the block is finished.

Warning:

When using function G08 it is not possible to execute a complete circle, as an infinite range of solutions exists. The CNC displays the corresponding error code.

6.6 ARC DEFINED BY THREE POINTS (G09)

Through function **G09** you can define an arc by programming the endpoint and an intermediate point (the starting point of the arc is the starting point of the movement). In other words, instead of programming the coordinates of the center, you program any intermediate point.

The endpoint of the arc is defined in Cartesian or polar coordinates, and the intermediate point is always defined in Cartesian coordinates by the letters I,J, or K, each one being associated to the axes as follows:

Axes X,U,A —> I
Axes Y,V,B —> J
Axes Z,W,C —> K

In Cartesian coordinates:

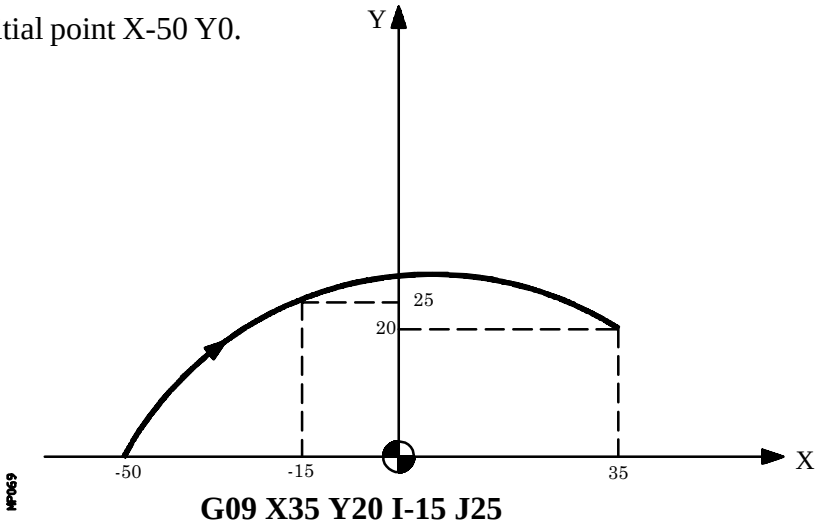
G17 **G09 X±5.5 Y±5.5 I±5.5 J±5.5**

Polar coordinates:

G17 **G09 R±5.5 Q±5.5 I±5.5 J±5.5**

Example:

Being initial point X-50 Y0.



G09 X35 Y20 I-15 J25

Function G09 is not modal, so it should always be programmed if you wish to execute an arc defined by three points. Function G09 can be programmed as G9.

When G09 is programmed it is not necessary to program the direction of movement (G02 or G03).

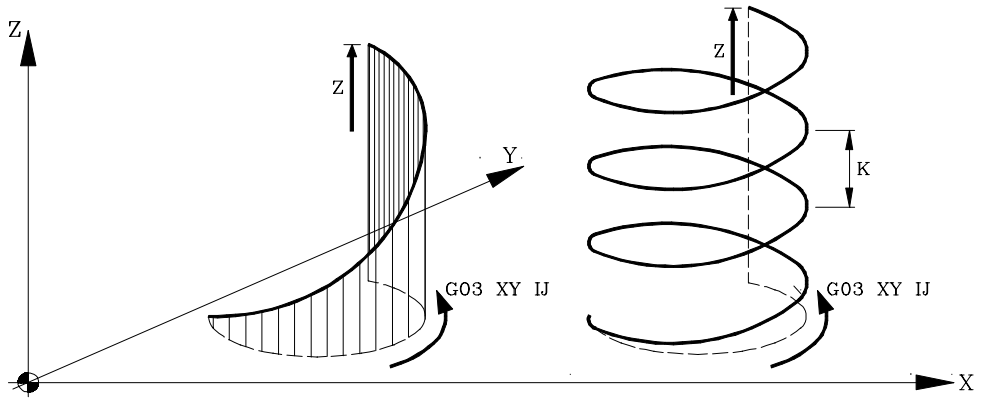
Function G09 does not alter the history of the program. The same G01, G02 or G03 function stays active after finishing the block.

Warning:

When using function G09 it is not possible to execute a complete circle, as you have to program three different points. The CNC displays the corresponding error code.

6.7 HELICAL INTERPOLATION

A helical interpolation consists in a circular interpolation in the work plane while moving the rest of the programmed axes.



The helical interpolation is programmed in a block where the circular interpolation must be programmed by means of functions: G02, G03, G08 or G09.

G02	X	Y	I	J		Z	G02	X	Y	R	Z	A
G03	Q	I	J	A		B	G08	X	Y	Z		
G09	X	Y	I	J		Z						

If the helical interpolation is supposed to make more than one turn, the linear movement of another axis must also be programmed (one axis only).

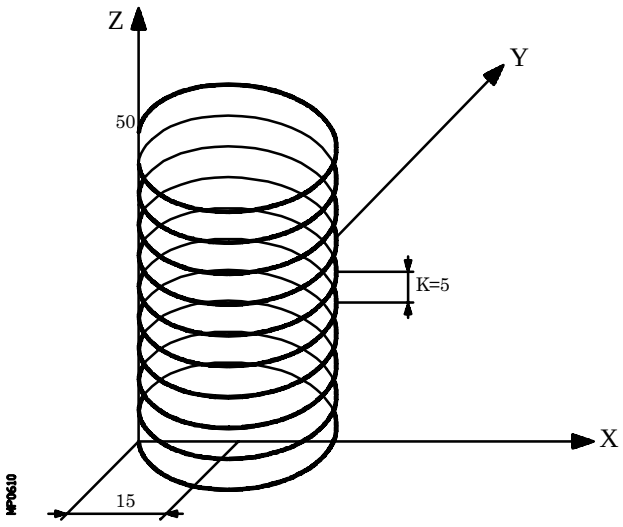
On the other hand, the pitch along the linear axis must also be set (format 5.5) by means of the I, J and K letters. Each one of these letters is associated with the axes as follows:

(I) for the X, U, A axes (J) for the Y, V, B axes (K) for the Z, W, C axes

G02	X	Y	I	J	Z	K	G02	X	Y	R	Z	K
G03	Q	I	J	A	I		G08	X	Y	B	J	
G09	X	Y	I	J	Z	K						

Example:

Programming in Cartesian and polar coordinates, the starting point being X0 Y0 Z0.



Cartesian coordinates:

G03 X0 Y0 I15 Z50 K5

Polar coordinates:

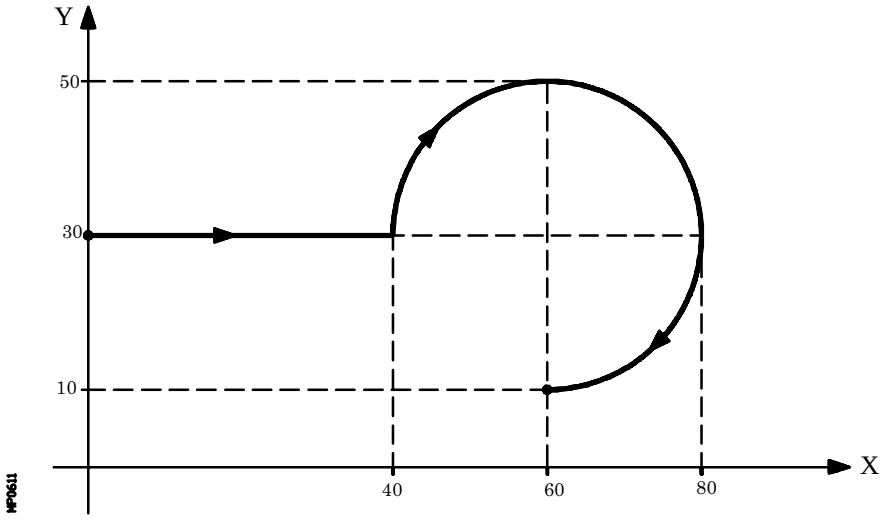
G03 Q180 I15 J0 Z50 K5

6.8 TANGENTIAL ENTRY AT BEGINNING OF A MACHINING OPERATION (G37)

Via function **G37** you can tangentially link two paths without having to calculate the intersection points.

Function G37 is not modal, so it should always be programmed if you wish to start a machining operation with tangential entry:

Example:

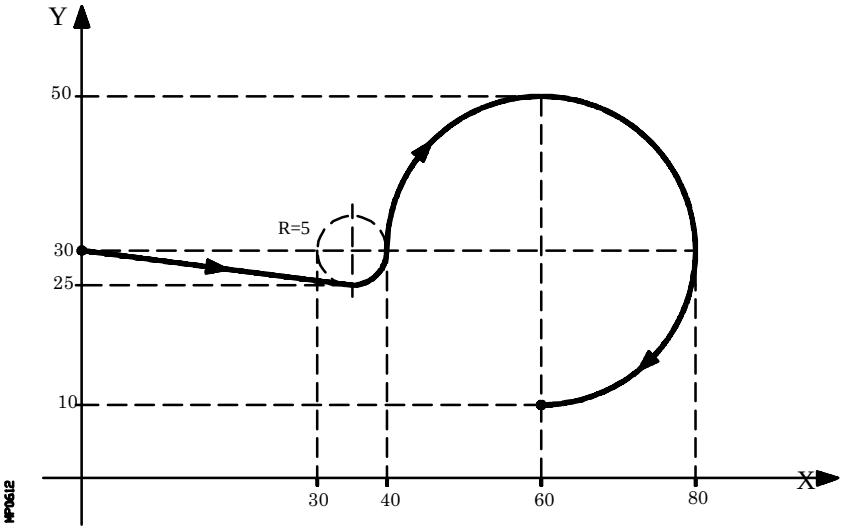


If the starting point is X0 Y30 and you wish to machine an arc (the path of approach being straight) you should program:

```
G90  G01 X40
     G02 X60 Y10 I20 J0
```

If, however, in the same example you require the entrance of the tool to the part to be machined tangential to the path and describing a radius of 5 mm, you should program:

```
G90  G01  G37  R5   X40
     G02  X60  Y10 I20  J0
```



As can be seen in the figure, the CNC modifies the path so that the tool starts to machine with a tangential entry to the part.

You have to program Function **G37** plus value **R** in the block which includes the path you want to modify.

R5.5 should appear in all cases following G37, indicating the radius of the arc which the CNC enters to obtain tangential entry to the part. Its value must always be positive.

Function G37 should only be programmed in the block which includes a straight-line movement (G00 or G01). If you program in a block which includes circular movement (G02 or G03), the CNC displays the corresponding error.

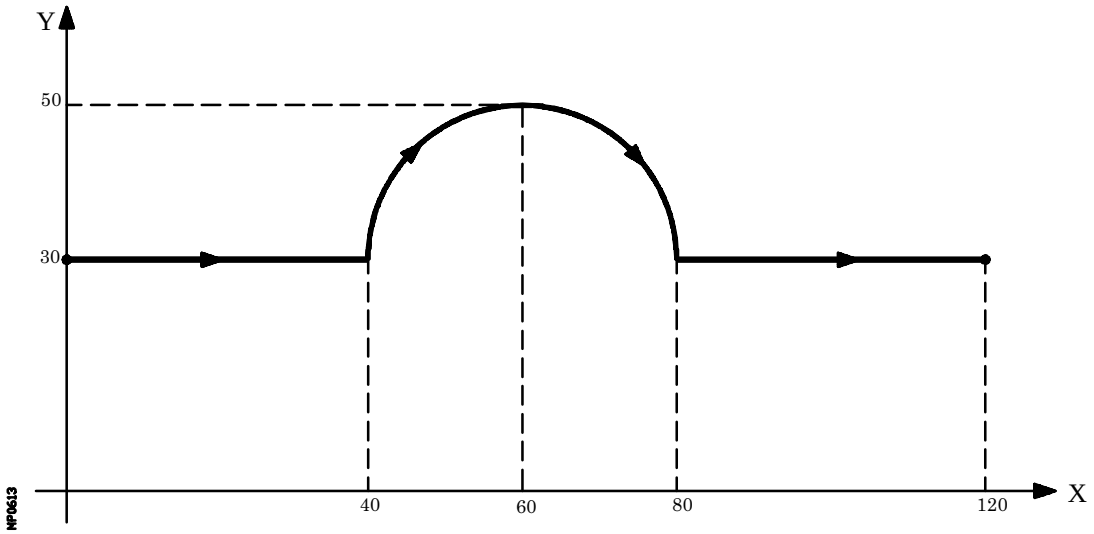
6.9 TANGENTIAL EXIT AT THE END OF A MACHINING OPERATION (G38)

Function **G38** enables the ending of a machining operation with a tangential exit of the tool. The path should be in a straight line (G00 or G01). Otherwise, the CNC will display the corresponding error.

Function G38 is not modal, so it should be programmed whenever a tangential exit of the tool is required.

Value **R 5.5** should always appear after G38. It also indicates the radius of the arc which the CNC applies to get a tangential exit from the part. This R value must always be positive.

Example:

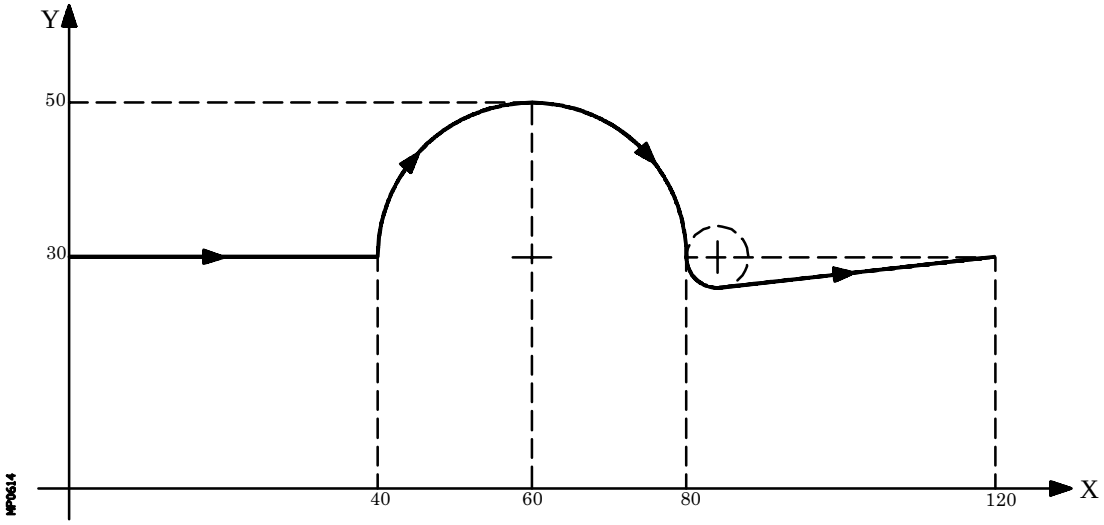


If the starting point is X0 Y30 and you wish to machine an arc (with the approach and exit paths in a straight line), you should program :

```
G90  G01  X40
     G02  X80   I20 J0
     G00  X120
```

If, however, in the same example you wish the exit from machining to be done tangentially and describing a radius of 5 mm, you should program :

```
G90  G01  X40
     G02  G38 R5 X80 I20 J0
     G00  X120
```



6.10 AUTOMATIC RADIUS BLEND (G36)

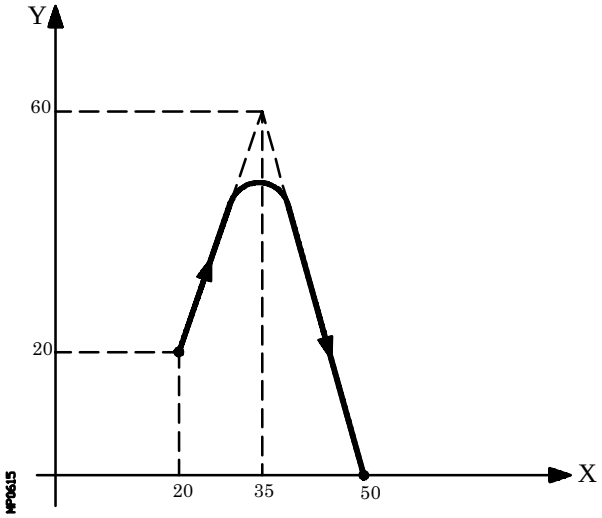
In milling operations, it is possible to round a corner via Function **G36** with a determined radius, without having to calculate the center nor the start and end points of the arc.

Function G36 is not modal, so it should be programmed whenever controlled corner rounding is required.

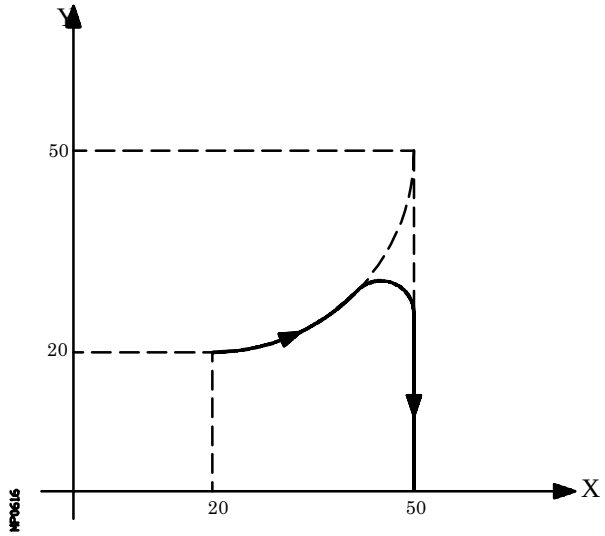
This function should be programmed in the block in which the movement the end you want to round is defined.

The **R5.5** value should always follow G36. It also indicates the rounding radius which the CNC applies to get the required corner rounding. This R value must always be positive.

Examples :



G90 G01 **G36** **R5** X35 Y60
X50 Y0



G90 G03 **G36** **R5** X50 I0 J30
G01 X50 Y0

6.11 AUTOMATIC CHAMFER BLEND (G39)

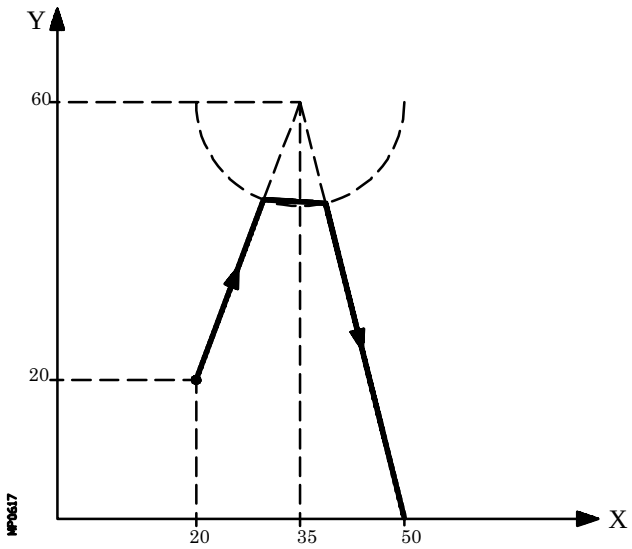
In machining operations it is possible (using G39) to chamfer corners between two straight lines, without having to calculate intersection points.

Function G39 is not modal, so it should be programmed whenever the chamfering of a corner is required.

This function should be programmed in the block in which the movement whose end you want to chamfer is defined.

The **R5.5** value should always follow G39. It also indicates the distance from the end of the programmed movement as far as the point where you wish to carry out the chamfering. This R value must always be positive.

Example :



G90 G01 **G39 R5** X35 Y60
X50 Y0

6.12 THREADING (G33)

If the machine spindle is equipped with a rotary encoder, you can thread with a tool tip via function **G33**.

Although this threading is often done along the entire length of an axis, the CNC enables threading to be done interpolating more than one axis at a time.

Programming format: **G33 X.....C L Q**

X...C ±5.5	End point of the thread
L5.5	Thread pitch
Q ±3.5	Optional. It indicates the spindle angular position (±359.9999) of the thread's starting point. If not programmed, a value of "0" is assumed.

Considerations:

Whenever G33 is executed and before making the thread, the CNC referenced the spindle (home search) and positions the spindle at the angular position indicated by parameter Q.

Parameter "Q" is available when spindle machine parameter "M19TYPE" has been set to "1".

If the threads are blended together in round corner, only the first one can have an entry angle (Q).

While function G33 is active, neither the programmed feedrate "F" nor the programmed Spindle speed "S" can be varied. They will both be set to 100%.

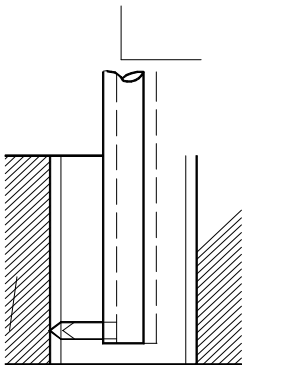
Function G33 is modal and incompatible with G00, G01, G02, G03 and G75.

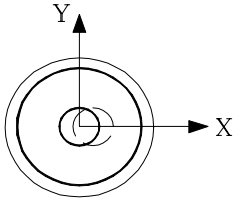
On power-up, after executing M02, M30 or after an EMERGENCY or RESET, the CNC assumes G00 or G01 depending on the setting of general machine parameter “IMOVE”

Example:

To make a 100mm deep and 5 mm pitch thread in a single pass at X0 Y0 Z0 with a threading tool located at Z10:

```
G90 G0 X Y Z    ; Positioning
G33  Z-100 L5   ; Threading
M19             ; Spindle orientation
G00 X3          ; Cutter withdrawal
Z30             ; Withdrawal (exit the hole)
```





6.13 MOVE TO HARDSTOP (G52)

By means of function G52 it is possible to program the movement of an axis until running into an object. This feature may be interesting for forming machines, live tailstocks, bar feeders, etc.

Its programming format is: **G52 X..C ±5.5**

After G52, program the desired axis as well as the target coordinate of the move.

The axis will move towards the programmed target coordinate until running into something. If the axis reaches the programmed target coordinate without running into the hardstop it will stop.

Function G52 is not modal; therefore, it must be programmed every time this operation is to be carried out.

Also, it assumes functions G01 and G40 modifying the program history.
It is incompatible with functions G00, G02, G03, G41, G42, G75 and G76.

6.14 FEEDRATE "F" AS AN INVERTED FUNCTION OF TIME (G32)

There are instances when it is easier to define the time required by the various axes of the machine to reach the target point instead of defining a common feedrate for all of them.

A typical case may be when a linear axis (X, Y, Z) has to move together (interpolated) with a rotary axis programmed in degrees.

Function G32 indicates that the "F" functions programmed next set the time it takes to reach the target point.

In order for a greater value of "F" to indicate a greater feedrate, the value assigned to "F" is defined as "Inverted function of time" and it is assumed as the activation of this feature.

"F" units: 1/min

Example: G32 X22 F4 indicates that the movement must be executed in ¼ minute. That is, in 0.25 minutes.

Function G32 is modal and incompatible with G94 and G95.

On power-up, after executing M02, M30 or after an Emergency or Reset, the CNC assumes G94 or G95 depending on the setting of general machine parameter "IFFED".

Considerations:

The CNC variable PROGFIN will show the feedrate programmed as an inverted function of time and variable FEED will show the resulting feedrate in mm/min or inches/min.

If the resulting feedrate of any axis exceeds the maximum value set by machine parameter "MAXFEED", the CNC will apply this maximum value.

The programmed "F" is ignored on G00 movements. All the movements will be carried out at the feedrate set by axis machine parameter "G00FEED".

When programming "F0" the movement will be carried out at the feedrate set by axis machine parameter “MAXFEED”.

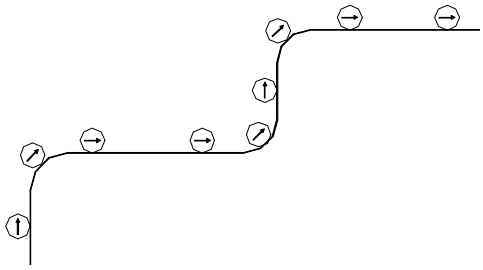
Function G32 may be programmed and executed in the PLC channel.

Function G32 is canceled in JOG mode.

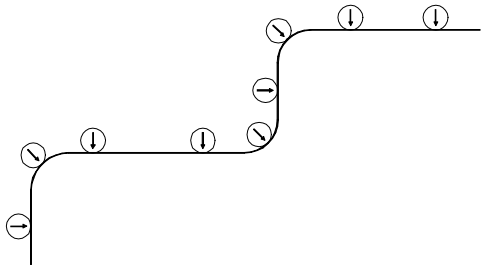
G32 is canceled when tracing. If it is programmed while tracing is active, the CNC will issue an error message.

6.15 TANGENTIAL CONTROL (G45)

With the "Tangential control" feature, the axis may maintain the same orientation with respect to the programmed path.



Orientation parallel to the path

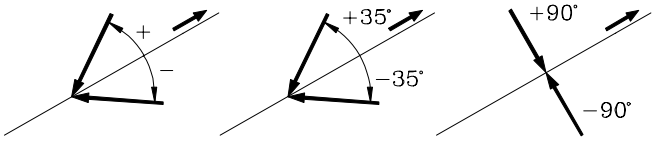


Orientation perpendicular to the path

The path is defined by the axes of the active plane. The axis maintaining the orientation must be a rotary rollover axis (A, B or C).

Programming format: **G45 Axis Angle**

Axis	axis maintaining the orientation (A, B or C)
Angle	Indicates the angular position in degrees with respect to the path (±359.9999). If not programmed, "0" will be assumed.



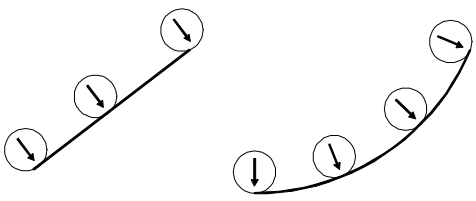
To cancel this function, program G45 alone (without defining the axis).

Every time G45 (tangential control) is activated, the CNC acts as follows:

1.- Positions the tangential axis, with respect to the first section in the programmed position.

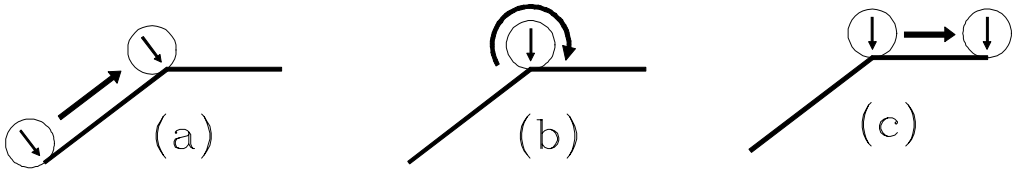
2.- The interpolation of the axes in the plane starts once the tangential axis has been positioned.

3.- On linear sections, the orientation of the tangential axis is maintained and in circular interpolations, the programmed orientation is maintained for the whole path.



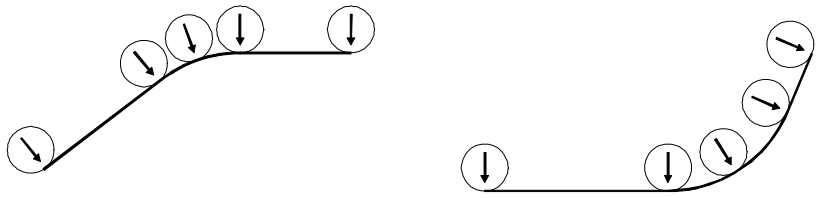
4.- If the joint of sections requires a new orientation of the tangential axis, the following takes place:

a) ends the current section.
b) orients the tangential axis with respect to the next section.
c) resumes execution.



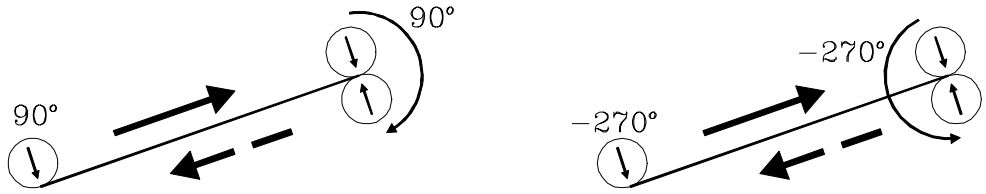
When working in round corner (G05), the tool orientation is not maintained at the corners since it begins before ending the current section.

It is recommended to work in square corner (G07). However, to work in round corner (G05), function G36 (automatic radius blend) should be used in order to also maintain tool orientation at the corners.



5.- To cancel the tangential control function, program G45 alone (without defining the axis).

Even when the tangential axis takes the same orientation by programming 90° or -270°, the turning direction in a direction change depends on the programmed value.



6.15.1 CONSIDERATIONS ABOUT FUNCTION G45

Tangential control, G45, is optional. It can only be executed in the main channel and is compatible with:

- Tool radius and length compensation (G40, 41, 42, 43, 44)
- Mirror image (G10, 11, 12, 13 14)
- Gantry axes , including the gantry axis associated with the tangential rotary axis.

The maximum feedrate while orienting the tangential axis is defined by machine parameter MAXFEED for that axis.

While tangential control is active, tool inspection is also possible. When accessing tool inspection, the tangential control is deactivated, the axes are free and when quitting tool inspection, tangential control may be activated again.

While in JOG mode, tangential control may be activated in MDI mode and the axes may be moved by programming blocks in MDI.

Tangential control is canceled when jogging the axes with the jog keys (not in MDI). Once the movement is over, tangential control is recovered.

On the other hand, the following is NOT possible:

- To define as tangential axis, one of the plane axes, the longitudinal axis or any other axis which is not rotary.
- To jog the tangential axis in JOG mode or by program using another G code while tangential control is active.
- Incline planes.

The TANGAN variable is read-only, from the CNC, PLC and DNC, associated with function G45. It indicates the angular position, in degrees, referred to the programmed path.

Also, general logic output TANGACT (M5558) indicates to the PLC that function G45 is active.

Function G45 is modal and is canceled when executing G45 alone (without defining the axis), on power-up, after executing an M02 or M30 or after an EMERGENCY or RESET.

7. ADDITIONAL PREPARATORY FUNCTIONS

7.1 INTERRUPTION OF BLOCK PREPARATION (G04)

The CNC reads up to 20 blocks ahead of the one it is executing, with the aim of calculating beforehand the path to be followed.

Each block is evaluated (in its absence) at the time it is read, but if you wish to evaluate it at the time of execution of the block you use function **G04**.

This function holds up the preparation of blocks and waits for the block in question to be executed in order to start the preparation of blocks once more.

A case in point is the evaluation of the "status of block-skip inputs" which is defined in the block header.

Example:

```
        .
        .
        .
    G04          ; interrupts block preparation
/1  G01 X10 Y20; block-skip condition "/1"
        .
        .
```

Function G04 is not modal, so it should be programmed whenever you wish to interrupt block preparation.

It should be programmed on its own and in the block previous to the one in which the evaluation in execution is required. Function G04 can be programmed as G4.

Every time G04 is programmed, active radius and length compensation are cancelled.

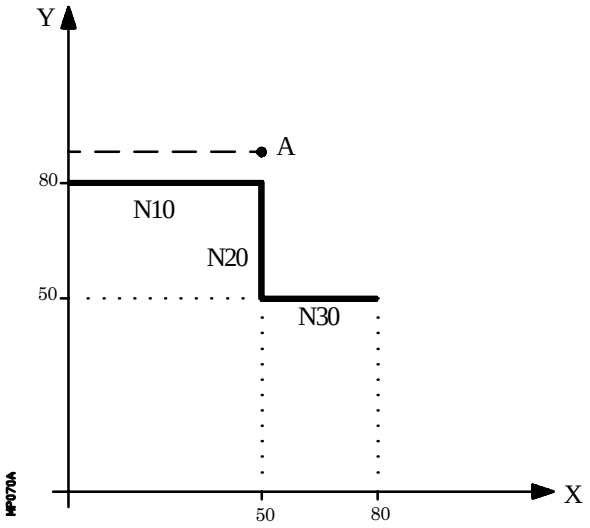
For this reason, care needs to be taken when using this function, because if it is introduced between machining blocks which work with compensation, unwanted profiles may be produced.

Example :

The following program blocks are executed in a section with G41 compensation :

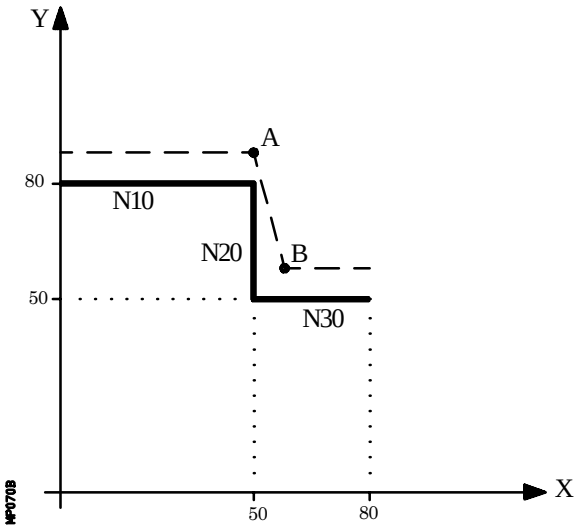
```
.....
.....
N10 X50 Y80
N15 G04
/1 N17 M10
   N20 X50 Y50
   N30 X80 Y50
.....
.....
```

Block N15 holds back the preparation of blocks so that the execution of block N10 ends up at point A.



Once the execution of block N15 has been carried out, the CNC continues preparing blocks starting from block N17.

Given that the next point corresponding to the compensated path is point “B”, the CNC moves the tool to this point, executing path “A-B”.



As you can see, the resulting path is not the required one, so we recommend avoiding the use of function G04 in sections which work with compensation.

7.2 DWELL (G04 K)

Timing can be programmed via function **G04 K**.

The timing value is programmed in hundredths of a second via format **K5 (0..99999).**

Example :

```
G04 K50  ; Timing of 50 hundredths of a second (0.5 seconds)
G04 K200 ; Timing of 200 hundredths of a second (2 seconds)
```

Function G04 K is not modal, so it should be programmed whenever timing is required. Function G04 K can be programmed as G4 K.

Timing is executed at the beginning of the block in which it is programmed.

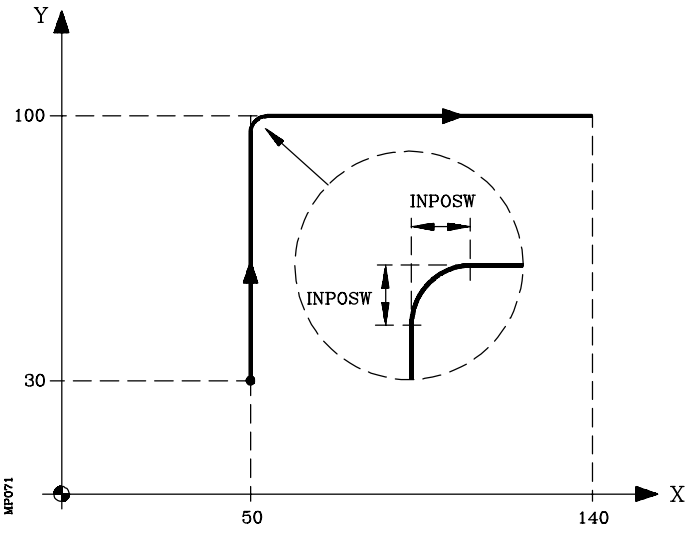
7.3 WORKING WITH SQUARE (G07) AND ROUND (G05,G50) CORNERS

7.3.1 SQUARE CORNER (G07)

When working in **G07** (square corner) the CNC does not start executing the following program block until the position programmed in the current block has been reached.

The CNC considers that the programmed position has been reached when the axis is within the "INPOSW" (in-position zone or dead band) from the programmed position.

Example:



G91 G01 **G07** Y70 F100
X90

The theoretical and real profile coincide, obtaining square corners, as seen in the figure.

Function G07 is modal and incompatible with G05 and G50. Function G07 can be programmed as G7.

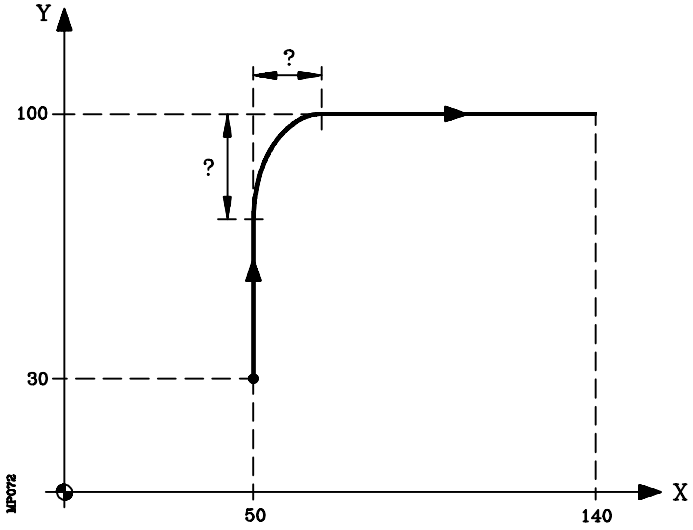
On power-up, after executing M02, M30 or after EMERGENCY or RESET, the CNC assumes code G05 or G07 depending on how the general machine parameter “ICORNER” is set.

7.3.2 ROUND CORNER (G05)

When working in **G05** (round corner), the CNC starts executing the following block of the program as soon as the theoretical interpolation of the current block has concluded. It does not wait for the axes to physically reach the programmed position.

The distance prior to the programmed position where the CNC starts executing the next block depends on the actual axis feedrate.

Example :



G91 G01 **G05** Y50 F100
X90

Via this function round corners can be obtained, as shown in the figure.

The difference between the theoretical and real profiles depends on the programmed feedrate value “F”. The higher the feedrate, the greater the difference between both profiles.

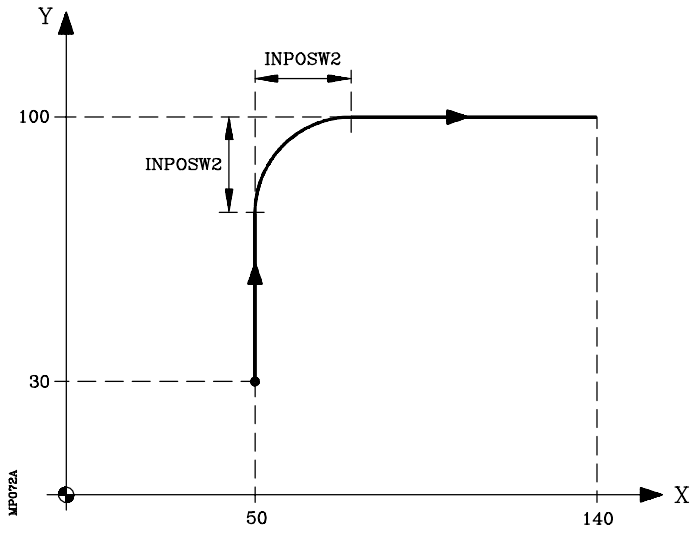
Function G05 is modal and incompatible with G07 and G50. Function G05 can be programmed as G5.

On power-up, after executing M02, M30 or after EMERGENCY or RESET, the CNC assumes code G05 or G07 depending on how the general machine parameter “ICORNER” is set.

7.3.3 CONTROLLED ROUND CORNER (G50)

When working in **G50** (controlled round corner); once the theoretical interpolation of the current block has concluded, the CNC waits for the axis to enter the area defined by machine parameter "INPOSW2" and it then starts executing the following block of the program.

Example :



G91 G01 **G50** Y50 F100
X90

Function G50 assures that the difference between the theoretical and actual paths stays smaller than what was set by machine parameter "INPOSW2".

On the other hand, when working in G05, the difference between the theoretical and real profiles depends on the programmed feedrate value “F”. The higher the feedrate, the greater the difference between both paths.

Function G50 is modal and incompatible with G07, G05 and G51.

On power-up, after executing M02, M30 or after EMERGENCY or RESET, the CNC assumes code G05 or G07 depending on how the general machine parameter “ICORNER” is set.

7.4 LOOK-AHEAD (G51)

Usually, a program consisting of very small movement blocks (CAM, digitizing, etc.) run very slowly.

With this feature, high speed machining is possible for this type of programs.

It is recommended to have the CPU-TURBO feature when using LOOK-AHEAD because the CNC has to analyze the machining path ahead of time (up to 50 blocks) in order to calculate the maximum feedrate for each section of the path.

The programming format is: G51 [A] E

A (0-255) Is optional and it defines the percentage of acceleration to be applied. When not programmed or programmed with a "0" value, the CNC assumes the acceleration value set by machine parameter for each axis.
E (5.5) Maximum contouring error allowed.

Parameter "A" permits using a standard working acceleration and another one to be used when executing with Look-Ahead.

The smaller the "E" parameter value, the lower the machining feedrate.

When operating with "Look-Ahead", it is a good idea to adjust the axes so their following error (lag) is as small as possible because the contouring error will be at least equal to the minimum following error.

When calculating the axis feedrate, the CNC takes into consideration the following aspects:

* The programmed feedrate.
* The curvature and the corners.
* The maximum feedrates of the axes.
* The maximum accelerations.

If any of the circumstances listed below occurs while executing with Look-Ahead, the CNC slows down to "0" at the previous block and it recovers the machining conditions for Look-Ahead in the next motion block.

* Motionless block.
* Execution of auxiliary functions (M, S, T).
* Single block execution mode.
* MDI mode.
* TOOL INSPECTION mode.

If a Cycle Stop, Feed-Hold, etc. occurs while executing in Look-Ahead mode, the machine may not stop at the current block, several additional blocks will be necessary to stop with the permitted deceleration.

Function G51 is modal and incompatible with G05, G07 and G50. Should any of them be programmed, function G51 will be canceled and the new one will be selected.

On the other hand, the CNC will issue Error 7 (Incompatible G functions) when programming any of the following functions while G51 is active:

* G23, G26, G27 Tracing
* G33 Electronic threading
* G52 Movement against hardstop
* G74 Home search
* G75, G76 Probing
* G95 Feedrate per revolution

Function G51 must be programmed alone in a block and there must be no more information in that block.

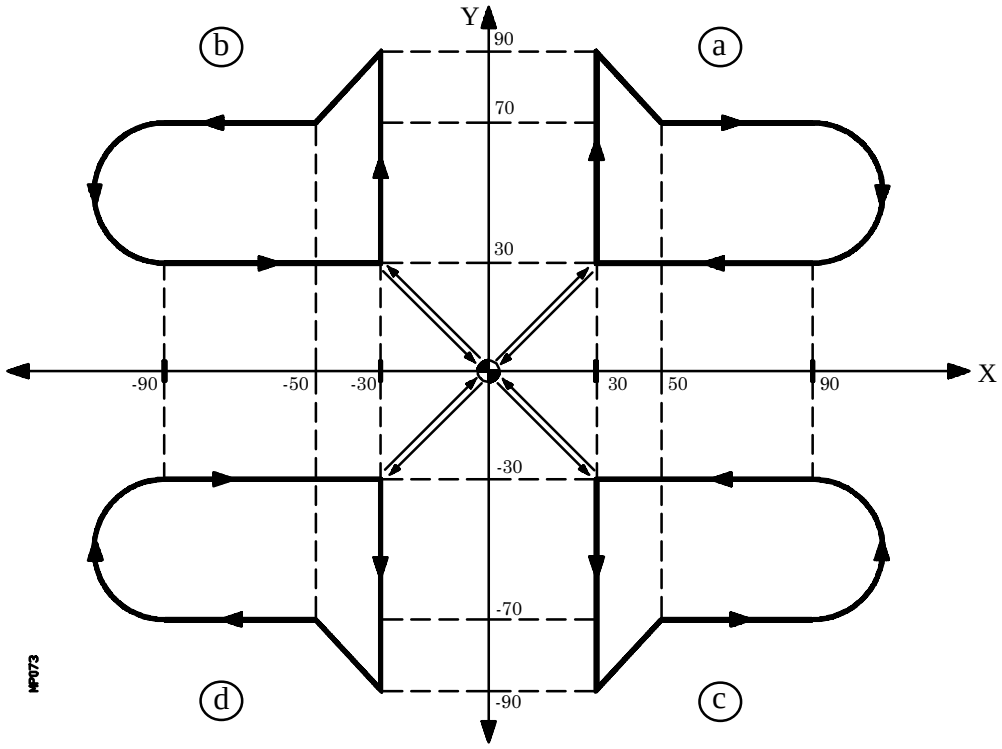
On power-up, after executing an M02, M30, of after an EMERGENCY or RESET, the CNC will cancel G51, if it was active, and it will assume G05 or G07 according to the setting of general machine parameter “ICORNER”.

7.5 MIRROR IMAGE (G10, G11. G12, G13, G14)

G10: cancel mirror image
G11: mirror image on X axis
G12: mirror image on Y axis
G13: mirror image on Z axis
G14: mirror image on any axis (X..C), or in several at the same time.
Examples : G14 W G14 X Z A B

When the CNC works with mirror images, it executes the movements programmed in the axes which have mirror image selected, with the sign changed.

Example :



The following subroutine defines the machining of part “a”.

```
G91 G01 X30  Y30  F100
        Y60
    X20  Y-20
    X40
    G02 X0   Y-40 I0     J-20
    G01 X-60
    X-30 Y-30
```

The programming of all parts would be :

```
Execution of subroutine  ; machines “a”
          G11            ; mirror image on X axis.
Execution of subroutine  ; machines “b
          G10 G12        ; mirror image on Y axis.
Execution of subroutine  ; machines “c”
          G11            ; mirror image on X and Y axes.
Execution of subroutine  ; machines “d”
          M30            ; end of program.
```

Functions G11, G12, G13, and G14 are modal and incompatible with G10.

G11, G12, and G13 can be programmed in the same block, because they are not incompatible with each other. Function G14 must be programmed alone in the block.

If function G73 (pattern rotation) is also active in a mirror image program, the CNC first applies the mirror image function and then the pattern rotation.

If while one of the mirror imaging functions (G11, G12, G13, and G14) is active, a new coordinate origin (part zero) is preset with G92, this new origin will not be affected by the mirror imaging function.

On power-up, after executing M02, M30 or after EMERGENCY or RESET, the CNC assumes code G10.

7.6 SCALING FACTOR (G72)

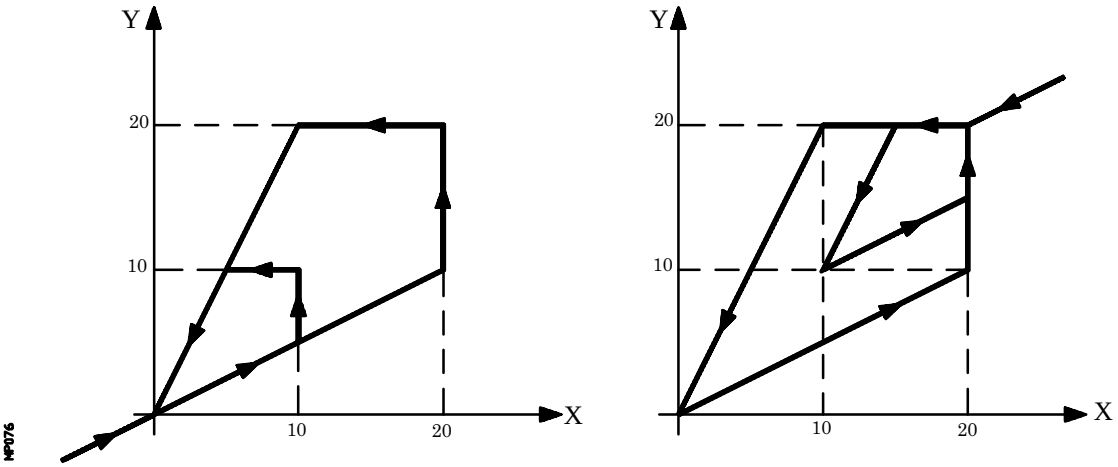
By using function **G72** you can enlarge or reduce programmed parts.

In this way, you can produce families of parts which are similar in shape but of different sizes with a single program.

Function G72 should be programmed on its own in a block. There are two formats for programming G72 :

Scaling factor applied to all axes.
Scaling factor applied to one or more axes.

Examples of application of the scaling factor.




```
        G90  G00    X0     Y0
N10     G91   G01   X20    Y10
                           Y10
                    X-10
N20                 X-10   Y-20
        G72 S0.5
        (RPT N10,20)

          M30
```

```
        G90   G00   X20    Y20
N10     G91   G01   X-10
                    X-10   Y-20
                    X20    Y10
N20                        Y10
        G72 S0.5           ; scaling factor
        (RPT N10,20)       ; repeats from
                           ; block 10
                           ; to block 20
          M30
```

Function G72 is modal and is cancelled when another scaling factor with a value of S1 is programmed, or on power-up, after executing M02, M30 or after EMERGENCY or RESET.

7.6.2 SCALING FACTOR APPLIED TO ONE OR MORE AXES

The programming format is :

G72 X...C 5.5

After **G72** the axis or axes and the required scaling factor are programmed.

All blocks programmed after G72 are treated by the CNC as follows :

The CNC calculates the movement of all the axes in relation to the programmed path and compensation.

It then applies the scaling factor indicated to the calculated movement of the corresponding axis or axes.

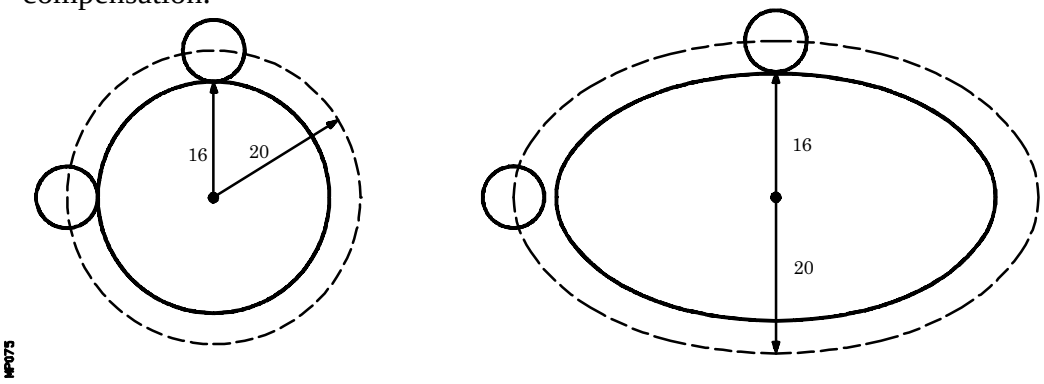
If the scaling factor is applied on one or more axes, the CNC will apply the scaling factor indicated both to the movement of the corresponding axis or axes and to their feedrate.

If, within the same program, both scaling factor types are applied, the one applied to all the axes and the one for one or several axes, the CNC applies a scaling factor equal to the product of the two scaling factors programmed for this axis to the axis or axes affected by both types.

Function G72 is modal and will be cancelled when the CNC is turned on, after executing M02, M30 or after an EMERGENCY or RESET.

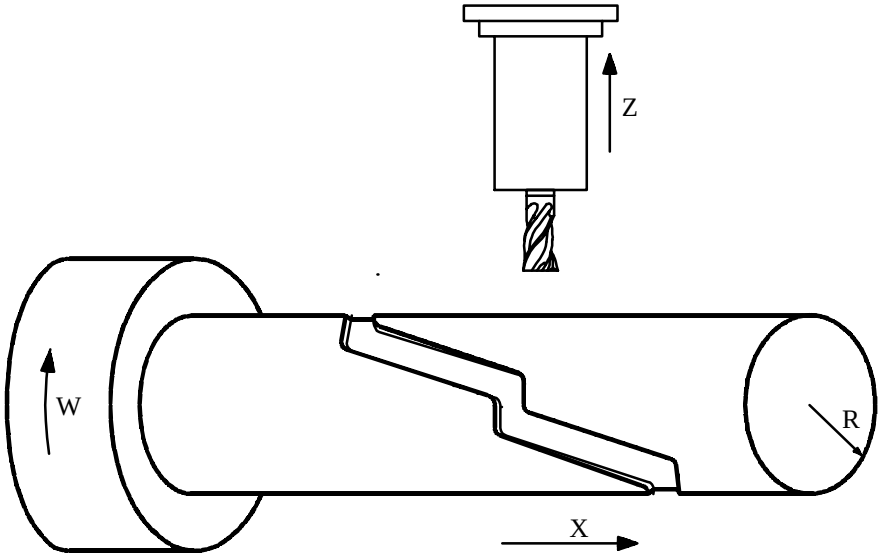
Example:

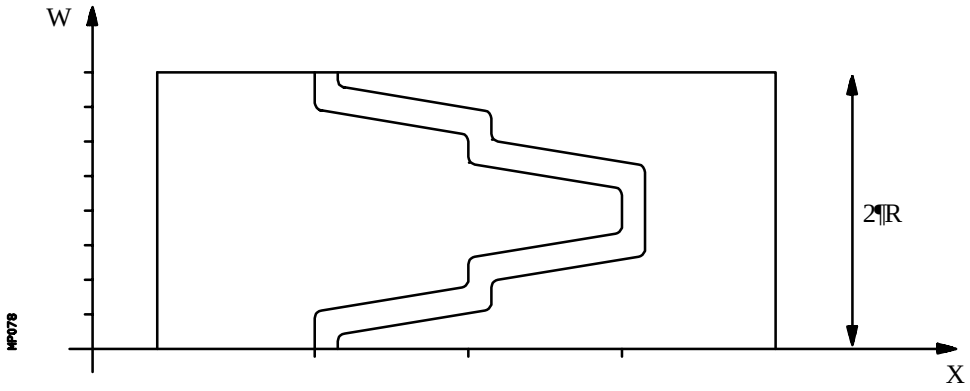
Application of the scaling factor to a plane axis, working with tool radius compensation.



As it can be observed, the tool path does not coincide with the required path, as the scaling factor is applied to the calculated movement.

However, if a scaling factor equal to 360/(2¶R) is applied to a rotary axis, R being the radius of the cylinder on which you wish to machine, this axis can be considered linear, and any figure with tool radius compensation can be programmed on the cylindrical surface.





7.7 PATTERN ROTATION (G73)

Function **G73** enables you to turn the system of coordinates, taking either the coordinates origin or the programmed rotation center as the active rotation center.

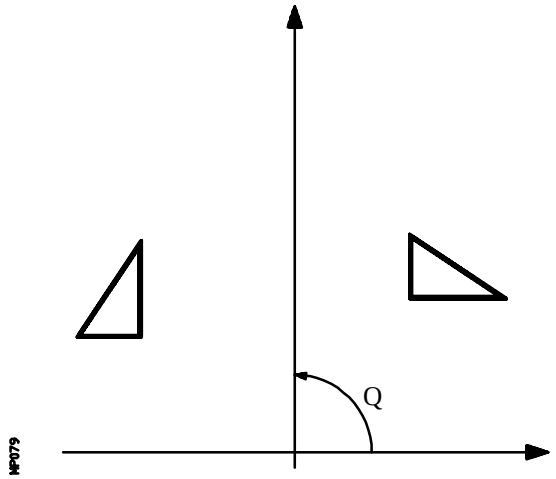
The format which defines the rotation is the following :

G 73 Q+/5.5 I±5.5 J±5.5

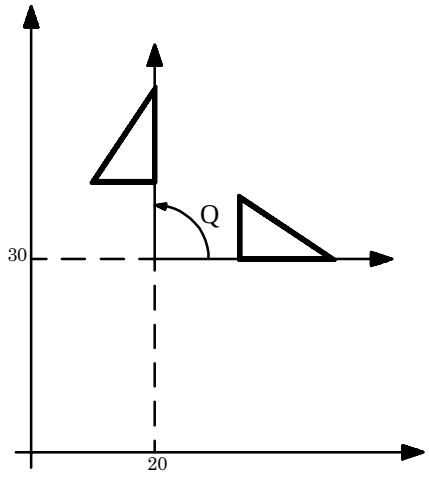
In which :

Q : indicates the angle of rotation in degrees
I,J : are optional and define the abscissa and ordinate respectively of the rotation center. If they are not defined, the coordinate origin will be taken as the rotation center.

Values I and J are defined in absolute coordinates and referred to the coordinate origin of the work plane. These coordinates are affected by the active scaling factor and mirror images.

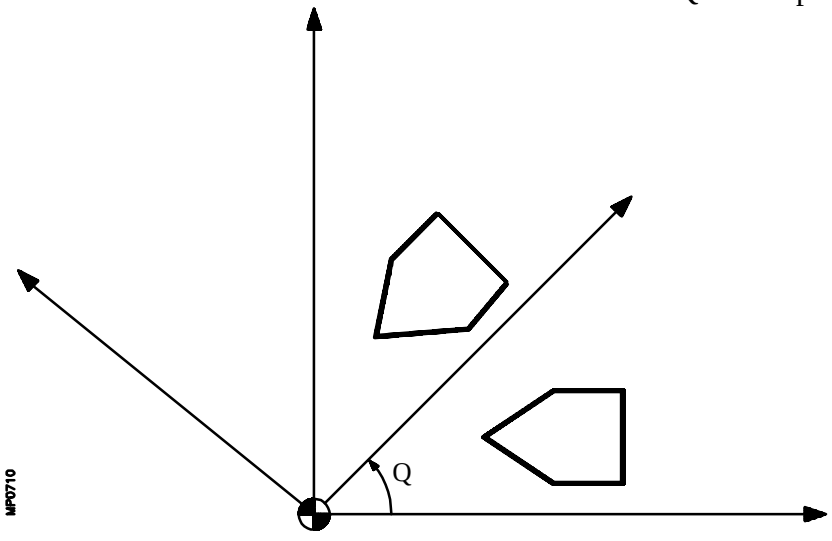


G73 Q90



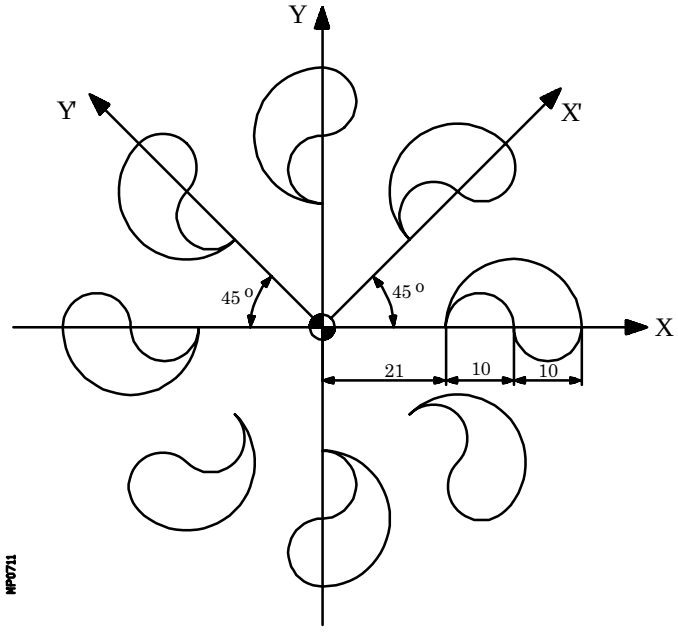
G73 Q90 I20 J30

You should remember that G73 is incremental i.e. the different Q values programmed add up.



Function G73 should be programmed on its own in a block.

Example :



Assuming that the starting point is X0 Y0, you get :

```
N10  G01  X21  Y0    F300  ; positioning at starting point
     G02  Q0   I5    J0
     G03  Q0   I5    J0
          Q180 I-10  J0
N20  G73  Q45                ; pattern rotation
     (RPT N10,20)    N7      ; repeat blocks 10 thru 20 seven times
     M30                     ; end of program
```

In a program which rotates the coordinate system, if any mirror image function is also active the CNC first applies the mirror image function and then the turn.

The pattern rotation function can be cancelled either by programming G72 (on its own, without angle value) or via G16, G17, G18, or G19, or on power-up, after executing M02, M30 or after EMERGENCY or RESET.

7.8 SLAVED AXIS/CANCELLATION OF SLAVED AXIS

The CNC enables two or more axes to be coupled together. The movement of all axes is subordinated to the movement of the axis to which they were coupled.

There are three possible ways of coupling axes :

Mechanical coupling. This is imposed by the manufacturer of the machine, and is selected via the axis machine parameter “GANTRY”.

By means of the PLC. This enables the coupling and uncoupling of each axis through logic input on the CNC “SYNCHRO1”, “SYNCHRO2”, “SYNCHRO3”, “SYNCHRO4”, and “SYNCHRO5”. Each axis is coupled to the one indicated in the axis machine parameter “SYNCHRO”.

By means of the program. This enables electronic coupling and uncoupling between two or more axes, through functions G77 and G78.

7.8.1 SLAVED AXIS (G77)

Function G77 allows the selection of both the master axis and the slaved axis (axes). The programming format is as follows :

G77 < Axis 1 > < Axis 2 > < Axis 3 > < Axis 4 > < Axis 5>

In which < Axis 2 > < Axis 3 > < Axis 4 > < Axis 5> indicate the slave axes you wish to couple to the master axis < Axis 1 >. You have to define < Axis 1 > and < Axis 2 >, the programming of the rest of the axes being optional.

Example :

```
G77 X Y U  ; couples Y and U axes to X axis
```

The following rules should be observed when doing electronic axis couplings :

You may use one or two different electronic couplings.

```
G77 X Y U  ; couples Y and U axes to X axis
G77 V Z    ; couples Z axis to V axis
```

You cannot couple one axis to two others at the same time.

```
G77 V Y    ; couples Y axis to V axis
G77 X Y    ; gives an error signal, because Y axis is coupled to V axis.
```

You can couple several axes to one in successive steps.

```
G77 X Z    ; couples Z axis to X axis
G77 X U    ; couples U axis to X axis —> Z U coupled to X
G77 X Y    ; couples Y axis to X axis —> Y Z U coupled to X
```

A pair of axes which are already coupled to each other cannot be coupled to another axis.

```
G77 Y U    ; couples U axis to Y axis
G77 X Y    ; gives an error signal, because Y axis is coupled to U axis.
```

7.8.2 SLAVED AXIS CANCELLATION (G78)

Function G78 enables you to uncouple all the axes which are coupled (slaved), or only uncouple indicated axes.

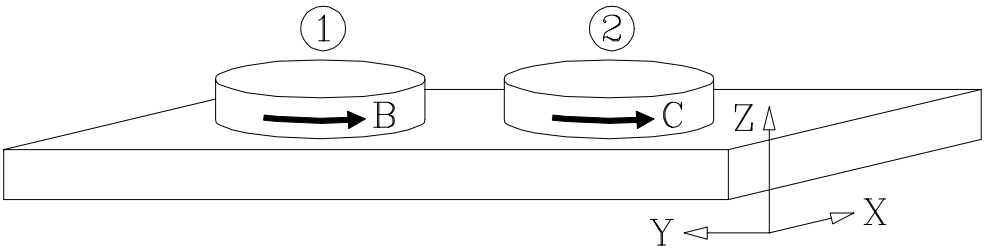
G78	Uncouples all slaved axes.
G78 <Axis 1><Axis 2><Axis 3><Axis 4>	Only uncouples indicated axes.

Example :

```
G77 X Y U ; slaves Y and U axes to X axis
G77 V Z   ; slaves Z axis to V axis
G78 Y     ; uncouples Y axis, but U stays slaved to X and Z to V.
G78       ; uncouples all axes.
```

7.9 AXES TOGGLE. G28-G29

With this feature, on machines having two machining tables, it is possible to use a single part-program to make the same parts on both tables.



With function G28 the axes can be toggled from one to the other in such way that after that instruction all the movements associated with the first axis next to G28 will take place on the second axis next to G28 and vice versa.

Programming format: G28 (axis 1) (axis 2)

To cancel the toggle, execute function G29 followed by one of the axes to be toggled back.

Up to three pairs of axes may be toggled at the same time.

The main axes cannot be toggled in the following cases: While tracing, while function G48 or G49 is active or when the "C" axis is active on a lathe.

On power-up, after executing an M30 or after an emergency or reset, the axes are toggled back as long as G48 or G49 is not active.

Example. Let us suppose that the part program is defined for table 1.

Execute the part-program on table 1	
G28 BC	Toggle the "B" and "C" axes
Zero offset	for machining on table 2
Execute the part-program	It will be executed on table 2
	In the meantime, replace the part made on table 1 with a new one
G29 B	Toggle the "B" and "C" axes back
Cancel the zero offset	for machining on table 1
Execute the part-program	It will be executed on table 1
	In the meantime, replace the part made on table 2 with a new one

8. *TOOL COMPENSATION*

The CNC has a tool offset table, its number of components being defined via the general machine parameter “NTOFFSET”. The following is specified for each tool offset :

* Tool radius, in work units, in R±5.5 format

* Tool length, in work units, in L±5.5 format.

* Wear of tool radius, in work units, in I±5.5 format. The CNC adds this value to the theoretical radius (R) to calculate the real radius (R+I).

* Wear of tool length, in work units, in K±5.5 format. The CNC adds this value to the theoretical length (L) to calculate the real length (L+K).

When tool radius compensation is required (G41 or G42), the CNC applies the sum of R+I values of the selected tool offset as the compensation value.

When tool length compensation is required (G43), the CNC applies the sum of L+K values of the selected tool offset as the compensation value.

8.1 TOOL RADIUS COMPENSATION (G40, G41, G42)

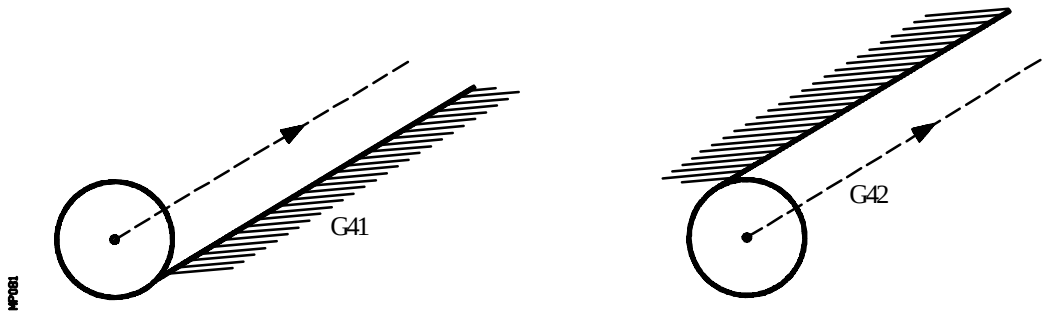
In normal milling operations, it is necessary to calculate and define the path of the tool taking its radius into account so that the required dimensions of the part are achieved.

Tool radius compensation allows the direct programming of part contouring and of the tool radius without taking the dimensions of the tool into account.

The CNC automatically calculates the path the tool should follow based on the contour of the part and the tool radius value stored in the tool offset table.

There are three preparatory functions for tool radius compensation:

G40 Cancelling of tool radius compensation
G41 Tool radius compensation to the left of the part.
G42 Tool radius compensation to the right of the part.



G41. The tool is to the left of the part, depending on the machining direction.

G42. The tool is to the right of the part, depending on the machining direction.

Tool values **R, L, I, K** should be stored in the tool offset table before starting machining, or should be loaded at the beginning of the program via assignments to variables **TOR, TOL, TOI, TOK.**

Once the plane in which compensation will be applied has been chosen via codes G16, G17, G18, or G19, this is put into effect by **G41** or **G42**, assuming the value of the tool offset selected via code **D**, or (in its absence) by the tool offset shown in the tool table for the selected tool (**T**).

Functions G41 and G42 are modal and incompatible to each other. They are cancelled by G40, G04 (interruption of block preparation), G53 (programming with reference to machine zero), G74 (home search), machining canned cycles (G81, G82, G83, G84, G85, G86, G87, G88, G89) and also on power-up, after executing M02, M30 or after EMERGENCY or RESET.

8.1.1 ACTIVATING TOOL RADIUS COMPENSATION

Once the plane in which tool radius compensation has been selected (via G16, G17, G18, or G19), functions G41 or G42 must be used to activate it.

G41 Compensation of tool radius compensation to the left.
G42 Compensation of tool radius compensation to the right.

In the same block (or a previous one) in which G41 or G42 is programmed, functions **T, D,** or only **T** must be programmed so that the tool offset value to be applied can be selected from the tool offset table. If no tool offset is selected, the CNC takes D0 with R0 L0 I0 K0.

When the new selected tool has an M06 associated to it and this M06, in turn, has a subroutine associated to it; the CNC will activate the tool radius compensation at the first movement block of that subroutine.

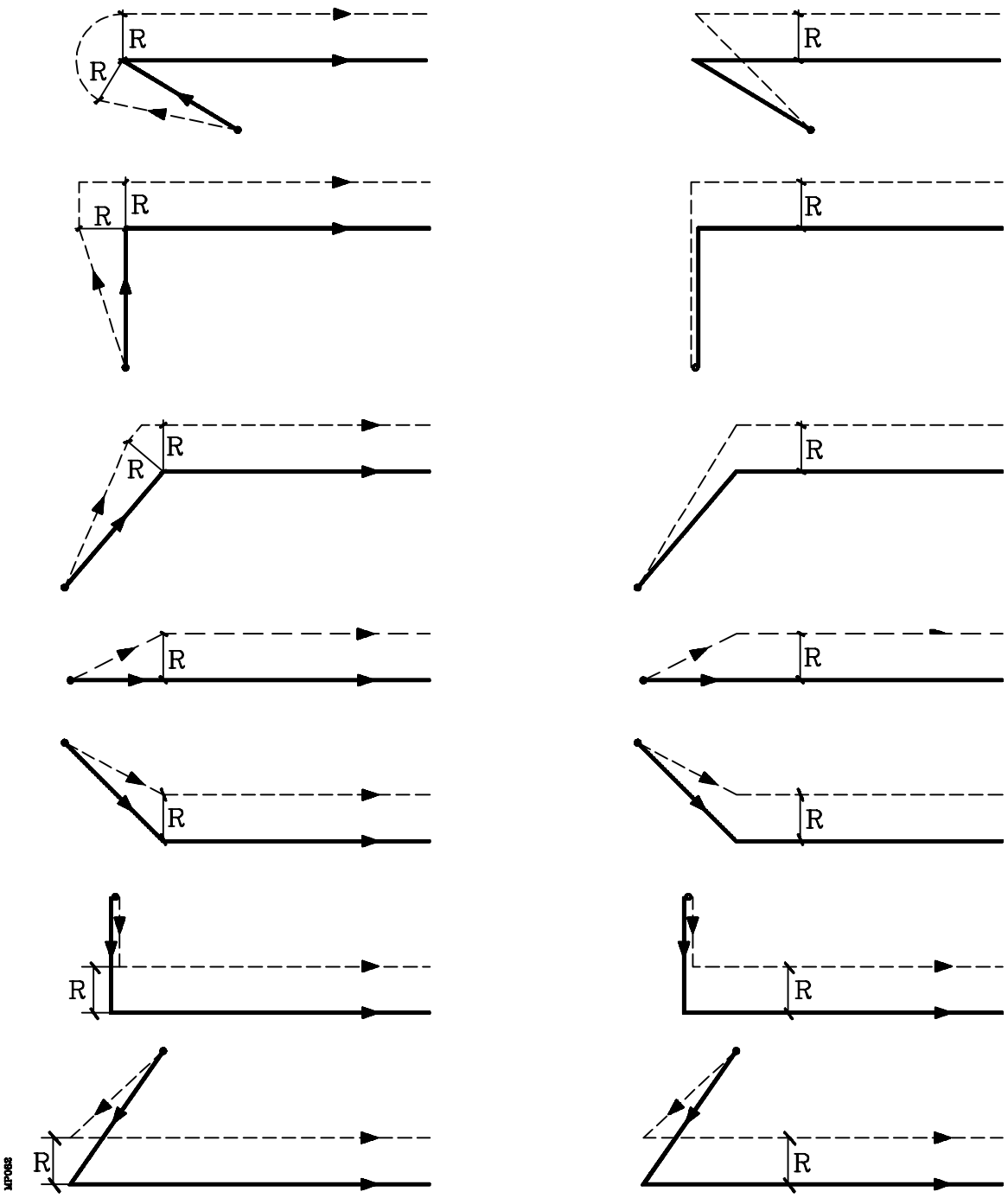
If that subroutine has a G53 programmed in a block (position values referred to Machine Reference Zero, home), the CNC will cancel any tool radius compensation (G41 or G42) selected previously.

The selection of tool radius compensation (G41 or G42) can only be made when functions G00 or G01 are active (straight-line movements).

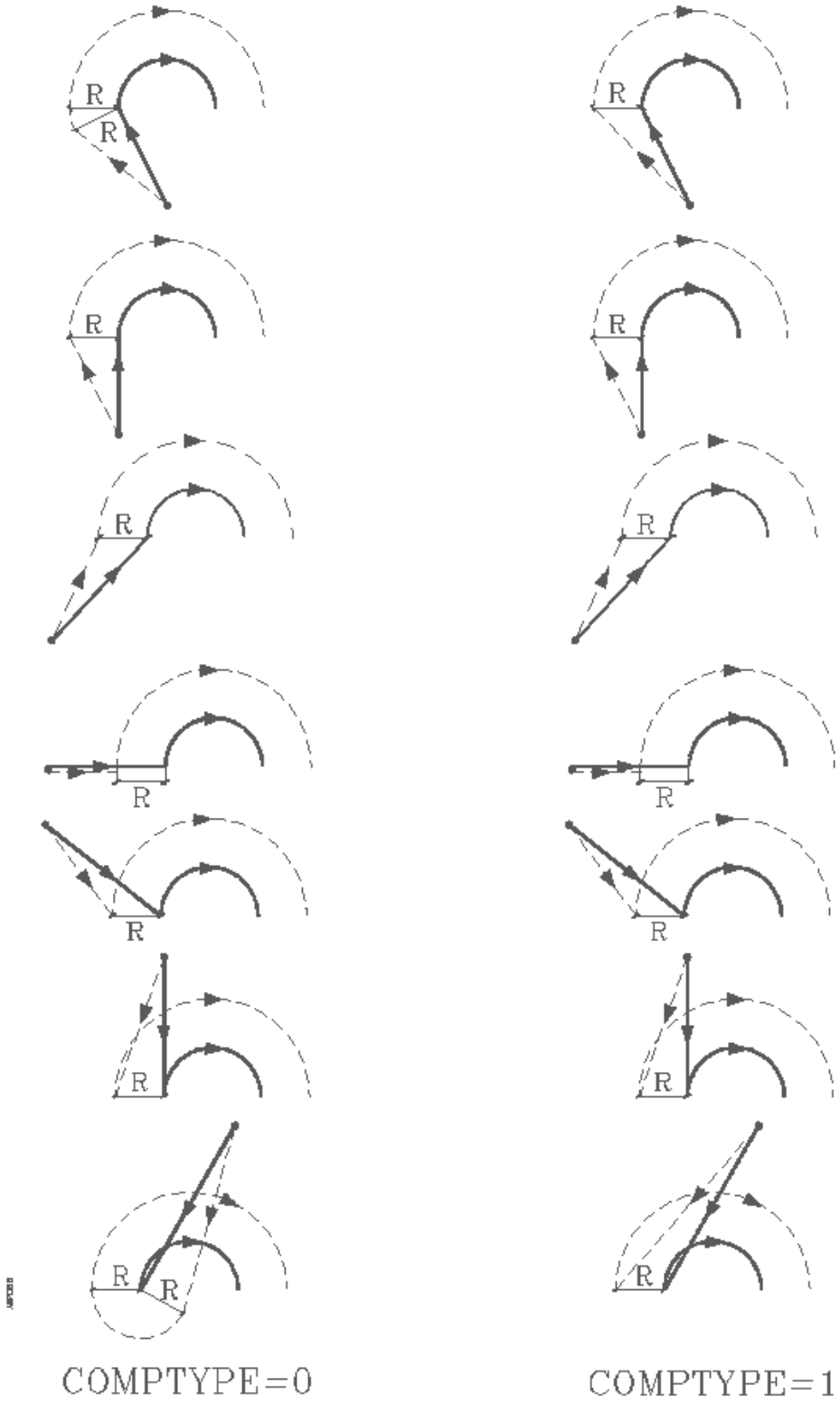
If the compensation is selected while G02 or G03 are active, the CNC will display the corresponding error message.

The following pages show different cases of starting tool radius compensation, in which the programmed path is represented by a solid line and the compensated path with a dotted line.

STRAIGHT-STRAIGHT path



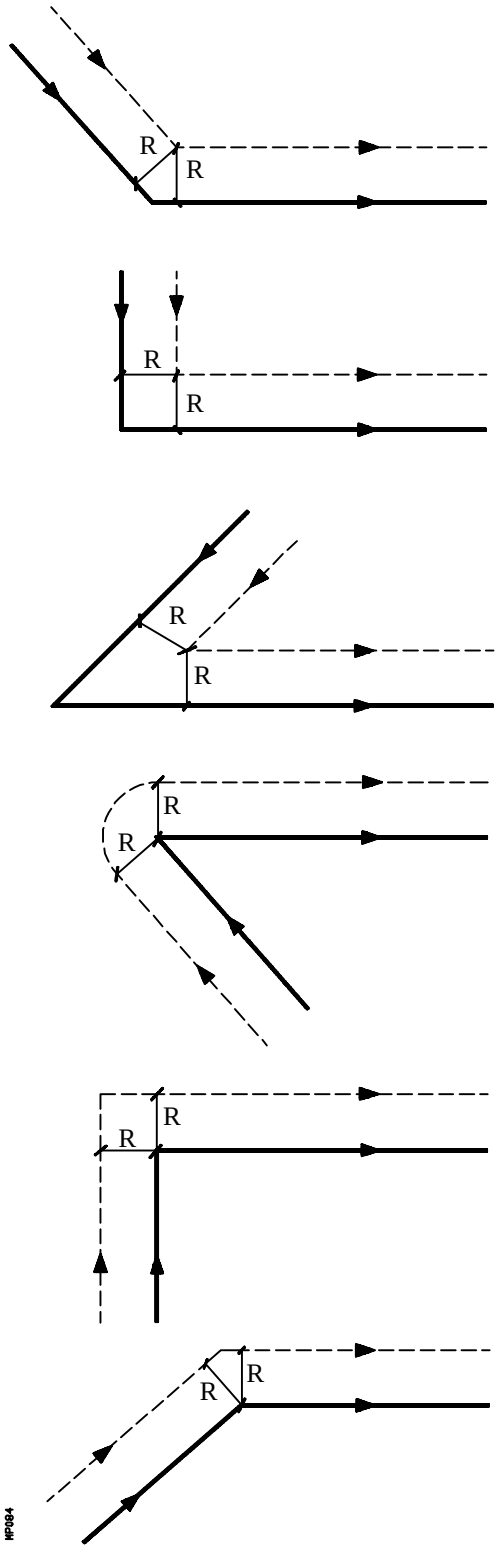
STRAIGHT-CURVED path

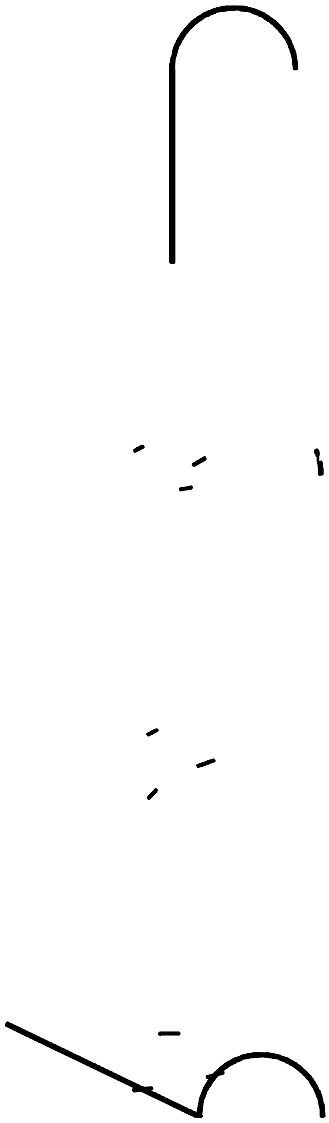


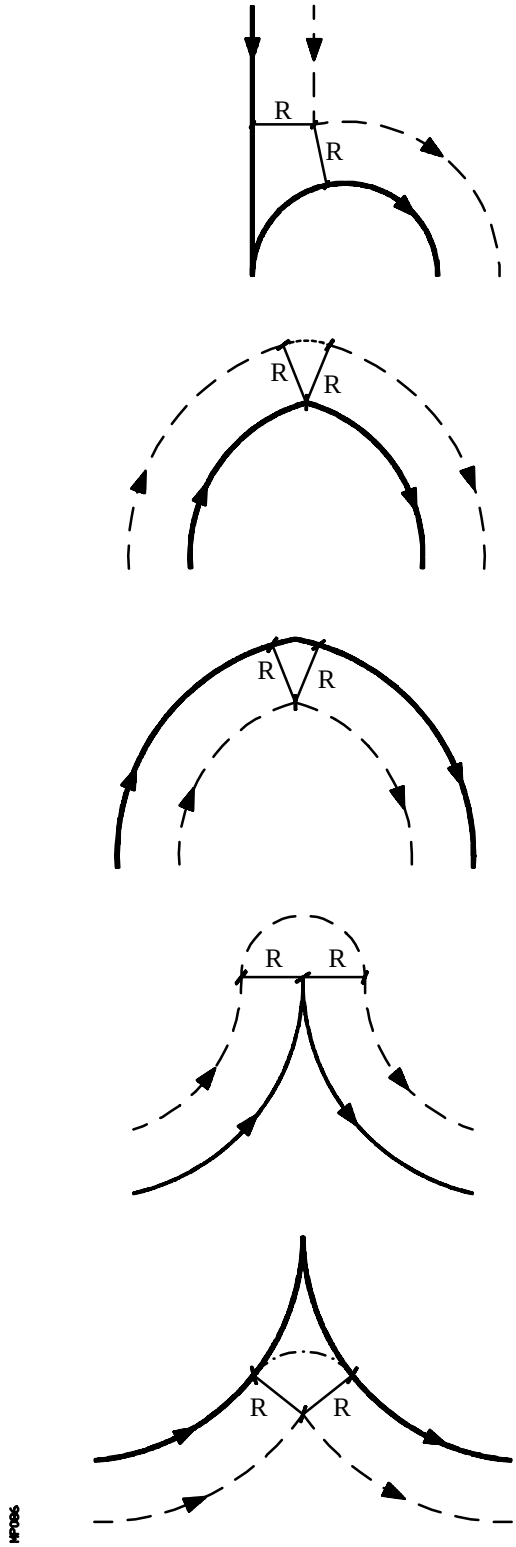
8.1.2 TOOL RADIUS COMPENSATION SECTIONS

The diagrams (below) show the different paths followed by a tool controlled by a programmed CNC with tool radius compensation.

The programmed path is represented by a solid line and the compensated path by a dotted line.






R
R
R
R
R
R
R
R
R
R
MP086

The CNC reads up to 20 blocks ahead of the one it is executing, with the aim of calculating in advance the path to be followed.

When the CNC works with compensation it needs to know the next programmed movement to calculate the path to be followed. For this reason, no more than 17 consecutive blocks can be programmed without movement.

8.1.3 CANCELLING TOOL RADIUS COMPENSATION

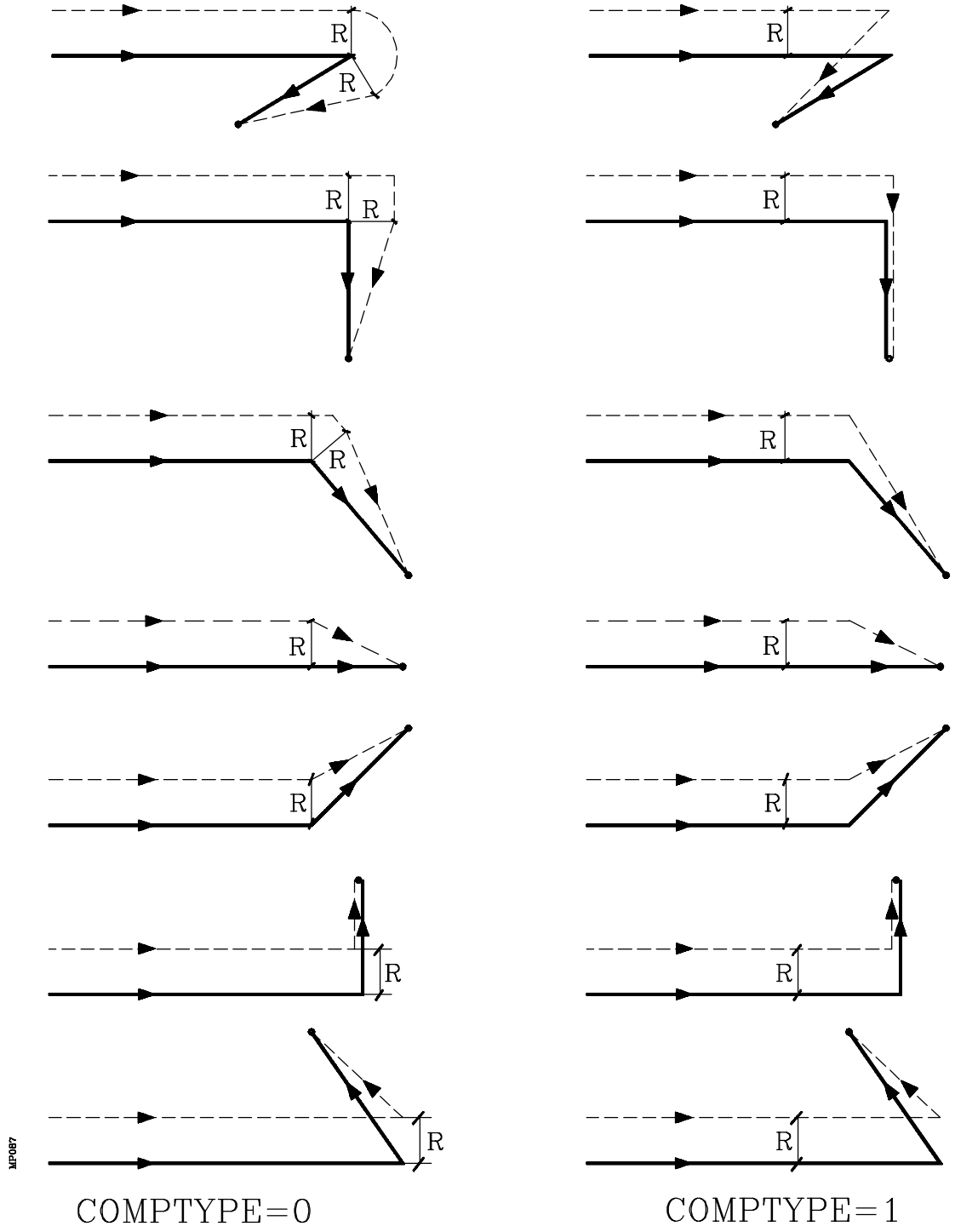
Tool radius compensation is cancelled by using function **G40**.

It should be remembered that cancelling radius compensation (G40) can only be done in a block in which a straight-line movement is programmed (G00 or G01).

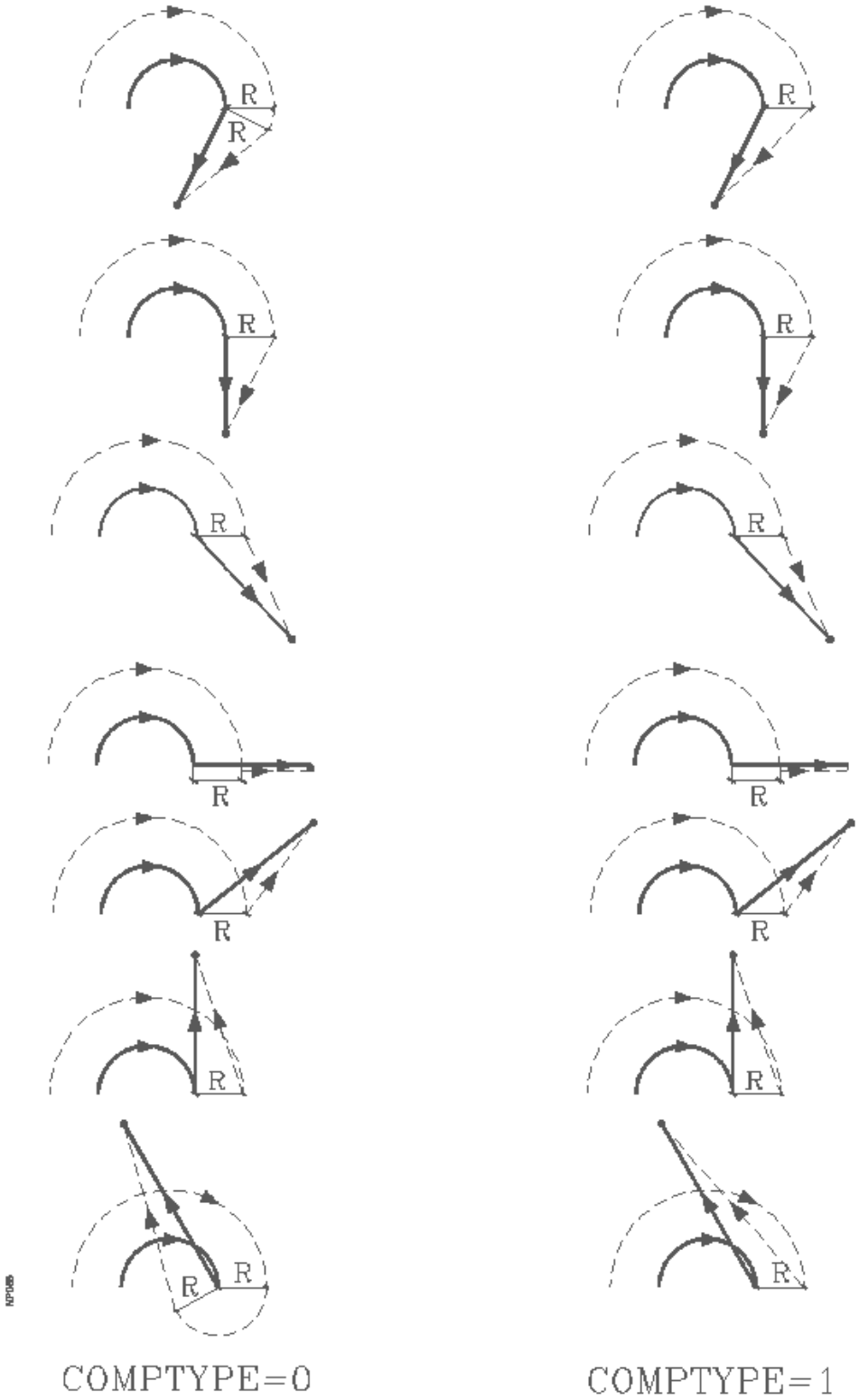
If G40 is programmed while functions G02 or G03 are active, the CNC displays the corresponding error message.

The following pages show different cases of cancelling tool radius compensation, in which the programmed path is represented by a solid line and the compensated path with a dotted line.

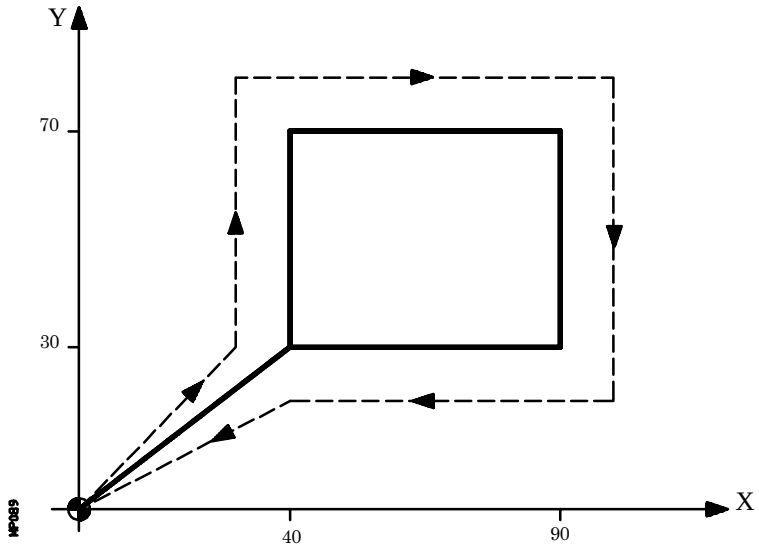
STRAIGHT-STRAIGHT path



CURVED-STRAIGHT path



Example of machining with radius compensation

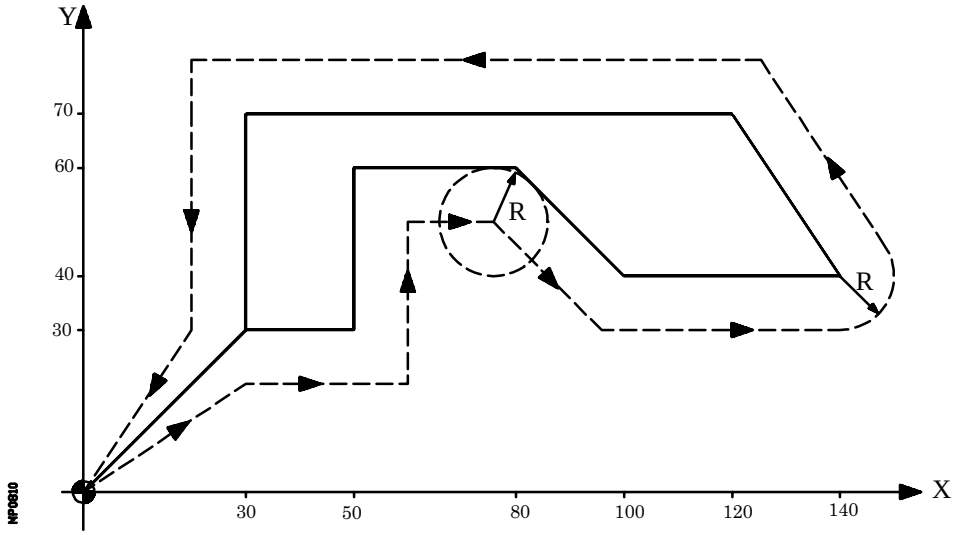


The programmed path is represented by a solid line and the compensation path by a dotted line.

Tool radius : 10mm.
Tool number : T1
Tool offset number : D1

```
G92        X0   Y0      Z0    ; position coordinate preset
G90  G17   S0.5 T1 D1   M03   ; tool, tool offset, spindle start at S100
G41  G01   X40  Y30     F125  ; activate compensation
                Y70
           X90
                Y30
           X40
G40  G01   X0   Y0            ; cancel compensation
M30
```

Example of machining with radius compensation :

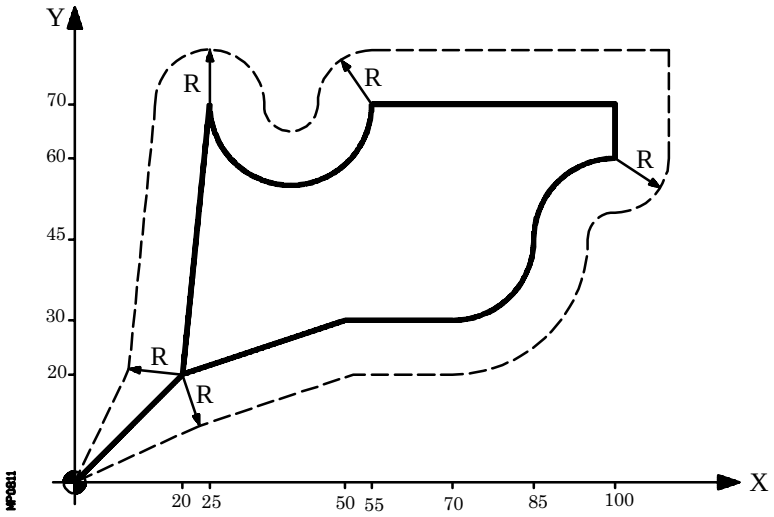


The programmed path is represented by a solid line and the compensation path by a dotted line.

Tool radius : 10mm.
Tool number : T1
Tool offset number : D1

```
G92  X0   Y0   Z0                          ; coordinate preset
G90  G17  G01  F150 S100 T1 D1 M03         ; tool, tool offset, spindle,..
G42  X30  Y30                              ; activate compensation
     X50
          Y60
     X80
     X100 Y40
     X140
     X120 Y70
     X30
          Y30
G40  G00  X0   Y0                          ; cancel compensation
M30
```

Example of machining with radius compensation :



The programmed path is represented by a solid line and the compensation path by a dotted line.

Tool radius : 10mm.
Tool number : T1
Tool offset number : D1

```
G92  X0   Y0   Z0                          ;  coordinate preset
G90  G17  G01  F150 S100  T1 D1 M03 ;  tool, tool offset, spindle,..
G42  X20  Y20                             ;  activate compensation
     X50  Y30
     X70
G03  X85  Y45  I0    J15
G02  X100 Y60  I15   J0
G01       Y70
     X55
G02  X25  Y70  I-15  J0
G01  X20  Y20
G40  G00  X0   Y0    M5                   ;  cancel compensation
M30
```

8.2 TOOL LENGTH COMPENSATION (G43, G44, G15)

With this function it is possible to compensate possible differences in length between the programmed tool and the tool being used.

The tool length compensation is applied on to the axis indicated by function G15 or, in its absence, to the axis perpendicular to the main plane.

If G17, tool length compensation on the Z axis.
If G18, tool length compensation on the Y axis
If G19, tool length compensation on the X axis.

Whenever one of functions G17, G18 or G19 is programmed, the CNC assumes as new longitudinal axis (upon which tool length compensation will be applied) the one perpendicular to the selected plane.

On the other hand, if function G15 is executed while functions G17, G18 or G19 are active, the new longitudinal axis (selected with G15) will replace the previous one.

The function codes used in length compensation are as follows:

G43 Activate tool length compensation.
G44 Cancelling tool length compensation.

Function G43 only indicates that a longitudinal compensation is to be applied. The CNC starts applying it when the longitudinal (perpendicular) axis starts moving.

```
Example:  G92 X0 Y0 Z50                        ; Preset
          G90 G17 G01 F150 S100 T1 D1 M03      ; Tool, Tool offset, etc.
          G43 X20 Y20                          ; Selects compensation
          X70
          Z30                                  ; Applies compensation
```

When G43 is programmed, the CNC compensates the length in accordance with the value of the tool offset selected with code **D**, or (in its absence) the tool offset shown in the tool table for the selected tool (**T**).

Tool values **R, L, I, K** must be stored in the tool offset table before starting machining, or must be loaded at the beginning of the program via assignments to variables **TOR, TOL, TOI, TOK.**

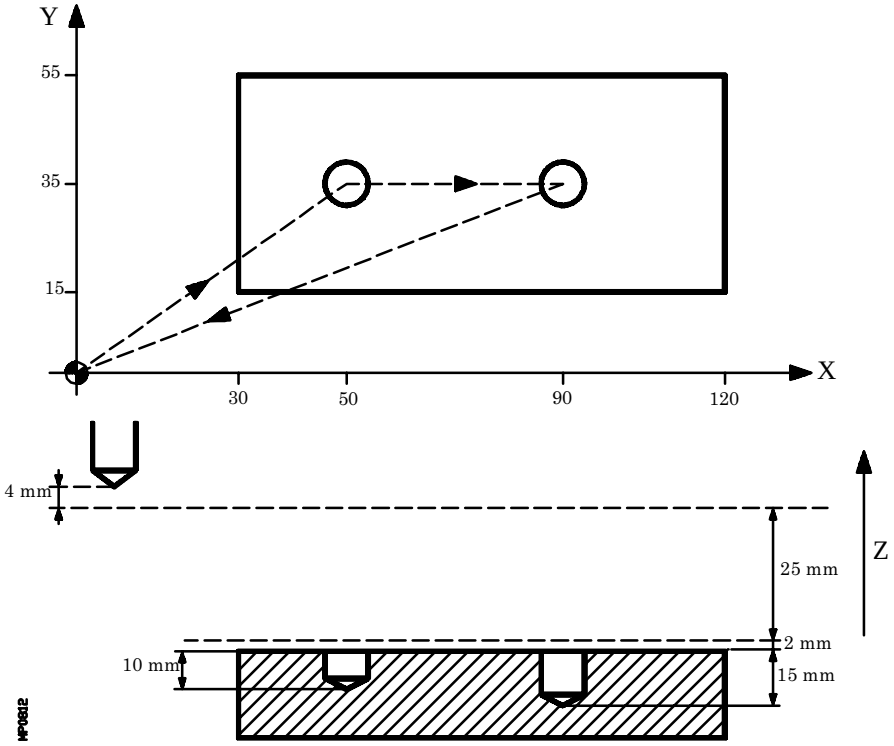
In the event of no tool offset being selected, the CNC takes D0 with values R0 L0 I0 K0.

Function G43 is modal and can be canceled via G44 and G74 (home search). If general machine parameter "ILCOMP=0", it is also canceled on power-up, after executing M02, M30 or after EMERGENCY or RESET.

G53 (programming with respect to machine zero) temporarily cancels G43 only while executing a block which contains a G53.

Length compensation can be used together with canned cycles, although here care should be taken to apply this compensation before starting the cycle.

Example of machining with length compensation :



It is assumed that the tool used is 4mm shorter than the programmed one.

Tool length : -4mm.
Tool number : T1
Tool offset number : D1

```
G92  X0   Y0   Z0                          ; coordinate preset
G91  G00  G05  X50  Y35  S500 M03
G43            Z-25            T1 D1       ; activate compensation
G01  G07       Z-12 F100
G00            Z12
               X40
G01            Z-17
G00  G05  G44  Z42  M05                     ; cancel compensation
G90  G07       X0   Y0
M30
```

9. *CANNED CYCLES*

These canned cycles can be performed on any plane, the depth being along the axis selected as longitudinal via function G15 or, in its absence, along the axis perpendicular to this plane.

The CNC offers the following machining canned cycles :

G69 Complex deep hole drilling

G81 Drilling cycle

G82 Drilling cycle with dwell

G83 Simple deep hole drilling

G84 Tapping cycle

G85 Reaming cycle

G86 Boring cycle with withdrawal in G00

G87 Rectangular pocket milling cycle

G88 Circular pocket milling cycle

G89 Boring cycle with withdrawal in G01

It also offers the following functions that can be used with the machining canned cycles:

G79 Modification of the canned cycle parameters
G98 Return to the starting plane at the end of the canned cycle
G99 Return to the reference plane at the end of the canned cycle.

9.1 DEFINITION OF A CANNED CYCLE

A canned cycle is defined by the G function indicating the canned cycle and its corresponding parameters.

A canned cycle cannot be defined in a block which has non-linear movements (G02, G03, G08, G09, or G33).

Also, a canned cycle cannot be executed while function G02, G03 or G33 is active. The CNC will issue the corresponding error message.

However, once a canned cycle has been defined in a block and following blocks, functions G02, G03, G08 or G09 can be programmed.

9.2 CANNED CYCLE AREA OF INFLUENCE

Once a canned cycle has been defined it remains active, and all blocks programmed after this block are under its influence while it is not cancelled.

In other words, every time a block is executed in which some axis movement has been programmed, the CNC will carry out (following the programmed movement) the machining operation which corresponds to the active canned cycle.

If, in a movement block within the area of influence of a canned cycle, the number of times a block is executed (repetitions) "N" is programmed at the end of the block, the CNC repeats the programmed positioning and the machining operation corresponding to the canned cycle the indicated number of times.

If a number of repetitions (times) "N0" is programmed, the machining operation corresponding to the **canned cycle will not be performed. The CNC will only carry out the programmed movement.**

If, within the area of influence of a canned cycle, there is a block which does not contain any movement, the machining operation corresponding to the defined canned cycle will not be performed, except in the calling block.

G81	Definition and execution of the canned cycle (drilling).
G90 G1 X100	The X axis moves to X100, where the hole is to be drilled.
G91 X10 N3	The CNC runs the following operation 3 times. * Incremental move to X10. * Runs the cycle defined above.
G91 X20 N0	Incremental move only to X20 (no drilling).

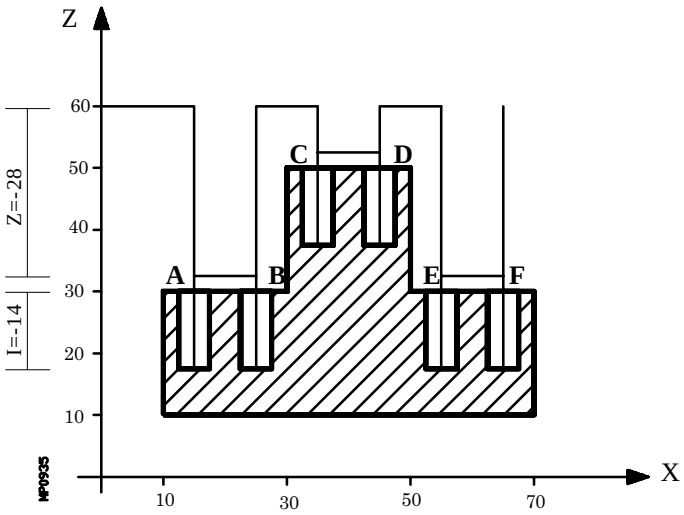
9.2.1. G79. MODIFICATION OF CANNED CYCLE PARAMETERS

The CNC allows one or several parameters of an active canned cycle to be modified by programming the G79 function, without any need for redefining the canned cycle. This is possible only inside the influence area of the canned cycle

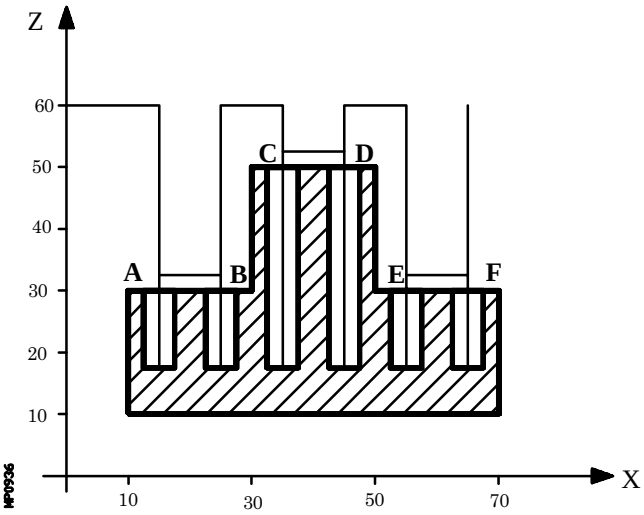
The CNC will continue to maintain the canned cycle active and will perform the following machinings of the canned cycle with the updated parameters.

The G79 function must be programmed alone in a block, and this block must not contain any more information.

Next 2 programming examples are shown assuming that the work plane is formed by the X and Y axes, and that the longitudinal axis (perpendicular) is the Z axis:




```
T1
M6
G00  G90  X0   Y0   Z60              ; Starting point
G81  G99  G91  X15  Y25  Z-28 I-14   ; Defines drilling cycle. Drills in A
G98  G90  X25                        ; Drills in B
G79  Z52                             ; Modifies reference plane and machining depth
G99  X35                             ; Drills in C
G98  X45                             ; Drills in D
G79  Z32                             ; Modifies reference plane and machining depth
G99  X55                             ; Drills in E
G98  X65                             ; Drills in F
M30
```




```
T1
M6
G00  G90  X0   Y0   Z60              ; Starting point
G81  G99  G90  X15  Y25  Z32  I18    ; Defines drilling cycle. Drills in A
G98  X25                             ; Drills in B
G79  Z52                             ; Modifies reference plane
G99  X35                             ; Drills in C
G98  X45                             ; Drills in D
G79  Z32                             ; Modifies reference plane
G99  X55                             ; Drills in E
G98  X65                             ; Drills in F
M30
```

9.3 CANNED CYCLE CANCELLATION

A canned cycle can be cancelled via :

- Function **G80**, which can be programmed in any block.
- After defining a new canned cycle. This will cancel and replace any other which may be active.
- After executing M02, M30, or after EMERGENCY or RESET.
- When searching home with function G74.
- Selecting a new work plane via functions G16, G17, G18, or G19.

9.4 GENERAL CONSIDERATIONS

1. A canned cycle can be defined at any point in a program, i.e., it can be defined both in the main program and in a subroutine.

2. Calls to subroutines can be made from a block within the influence of a canned cycle without implying the cancellation of the canned cycle.

3. The execution of a canned cycle will not alter the history of previous "G" functions.

4. Nor will the spindle turning direction be altered. A canned cycle can be entered with any turning direction (M03 or M04), leaving in the same direction in which the cycle was entered.

   Should a canned cycle be entered with the spindle stopped, it will start in a clockwise direction (M03), and maintain the same turning direction until the cycle is completed.

5. Should it be required to apply a scaling factor when working with canned cycles, it is advisable that this scale factor be common to all the axes involved.

6. The execution of a canned cycle cancels radius compensation (G41 and G42). It is equivalent to G40.

7. If tool length compensation (G43) is to be used, this function must be programmed in the same block or in the one before the definition of the canned cycle.

   The CNC applies the tool length compensation when the longitudinal (perpendicular) axis starts moving. Therefore, it is recommended to position the tool outside the canned cycle area when defining function G43 for the canned cycle.

8. The execution of any canned cycle will alter the global parameter **P299**.

9.5 MACHINING CANNED CYCLES

In all machining cycles there are three coordinates along the longitudinal axis to the work plane which, due to their importance, are discussed below:

Initial plane coordinate. This coordinate is given by the position which the tool occupies with respect to machine zero when the cycle is activated.

Reference plane coordinate. This is programmed in the cycle definition block and represents an approach coordinate to the part. It can be programmed in absolute coordinates or in incremental, in which case it will be referred to the initial plane.

Machining depth coordinate. This is programmed in the cycle definition block. It can be programmed in absolute coordinates or in incremental coordinates, in which case it will be referred to the reference plane.

There are two functions which allow to select the type of withdrawal of the longitudinal axis after machining.

G98 Selects the withdrawal of the tool as far as the initial plane, once the indicated machining has been done.

G99 Selects the withdrawal of the tool as far as the reference plane, once the indicated machining has been done.

These functions can be used both in the cycle definition block and the blocks which are under the influence of the canned cycle. The initial plane will always be the coordinate which the longitudinal axis had when the cycle was defined.

The structure of a canned cycle definition block is as follows:

G**	Starting point	Parameters	F S T D M	N****

It is possible to program the starting point in the canned cycle definition block (except the longitudinal axis), both in polar coordinates and in cartesian coordinates.

After defining the point at which it is required to carry out the canned cycle (optional), the functions and parameters corresponding to the canned cycle will be defined, and afterwards, if required, the complementary functions F S T D M are programmed.

If a number of block repetitions is programmed, the CNC will repeat the programmed positionings and the canned cycle machining operations the indicated number of times.

When programming, at the end of the block, the number of times a block is to be executed "N", the CNC performs the programmed move and the machining operation corresponding to the active canned cycle the indicated number of times.

If "N0" is programmed, it will not execute the machining operation corresponding to the canned cycle. The CNC will only execute the programmed move.

The general operation for all the cycles is as follows:

* If the spindle was in operation previously, its turning direction is maintained. If it was not in movement, it will start by turning clockwise (M03).

* Positioning (if programmed) at the starting point for the programmed cycle.

* Rapid movement of the longitudinal axis from the initial plane to the reference plane.

* Execution of the programmed machining cycle.

* Rapid withdrawal of the longitudinal axis to the initial plane or reference plane, depending on whether G98 or G99 has been programmed.

Below, a detailed explanation is given of machining canned cycles, assuming in all cases that the work plane is made up of the X and Y axes and that the longitudinal axis is the Z axis.

9.5.1 G69. COMPLEX DEEP HOLE DRILLING CYCLE

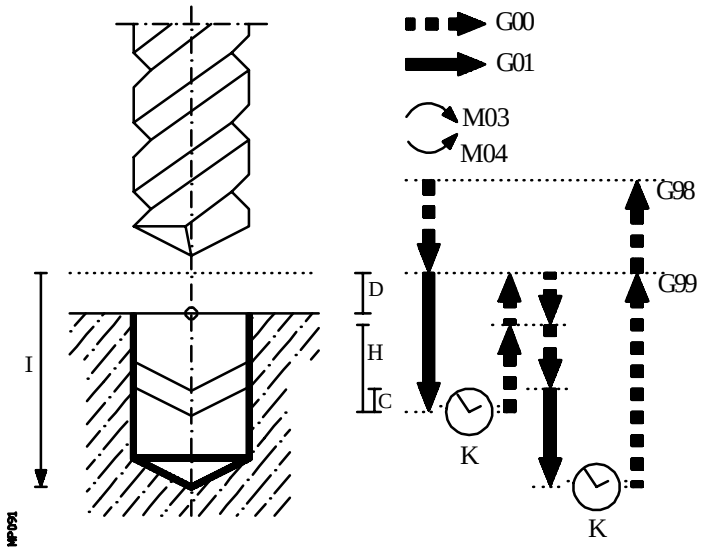
This cycle makes successive drilling steps until the final coordinate is reached.

The tool withdraws a fixed amount after each drilling operation, it being possible to select that every J drillings it withdraws to the reference plane.

A dwell can also be programmed after every drilling.

Working in cartesian coordinates, the basic structure of the block is as follows:

G69 G98/G99 X Y Z I B C D H J K L R



G98 The tool withdraws to the Initial Plane, once the hole has been drilled.

G99 The tool withdraws to the Reference Plane, once the hole has been drilled.

XY±5.5 These are optional and define the movement of the axes of the main plane to position the tool at the machining point.

This point can be programmed in cartesian coordinates or in polar coordinates, and the coordinates may be absolute or incremental, according to whether the machine is operating in G90 or G91.

Z±5.5 Defines the reference plane coordinate. It can be programmed in absolute coordinates or incremental coordinates, in which case it will be referred to the initial plane.

If this is not programmed, the CNC will take the position occupied by the tool at that moment as the reference plane.

I±5.5 Defines the total drilling depth. It can be programmed in absolute coordinates or incremental coordinates and in this case will be referred to the reference plane.

B5.5 Defines the drilling step in the axis longitudinal to the main plane.

C5.5 Defines to what distance from the previous drilling step, the longitudinal axis will travel in rapid feed (G00) in its approach to the part to make another drilling step.

If this is not programmed, the value of 1 mm (0.040 inch) will be taken. If programmed with a value of 0, the CNC will display the corresponding error.

D5.5 Defines the distance between the reference plane and the surface of the part where the drilling is to be done.

In the first drilling, this amount will be added to "B" drilling step. If it is not programmed, a value of 0 will be taken.

H5.5 Distance which the longitudinal axis will withdraw in rapid (G00) after each drilling step.

If this is not programmed, the longitudinal axis will withdraw to the reference plane. If programmed with a value of 0, the CNC will display the corresponding error.

J4 Defines after how many drilling steps the tool withdraws to the reference plane in G00. A value of between 0 and 9999 can be programmed.

If this is not programmed or is programmed with a value of 0, a value of 1 will be taken, i.e., it will return to the reference plane after each drilling step.

K5 Defines the dwell time, in hundredths of a second, after each drilling step, until the withdrawal begins. Should this not be programmed, the CNC will take a value of K0.

L5.5 Defines the minimum value which the drilling step can acquire. This parameter is used with R values other than 1mm (0.040 inch). If this is not programmed or programmed with a value of 0, a value of 1 will be taken.

R5.5 Factor which reduces the drilling step "B". If this is not programmed or programmed with a value of 0, a value of 1 will be taken.

If R equals 1, all the drilling steps will be the same and the programmed value "B".

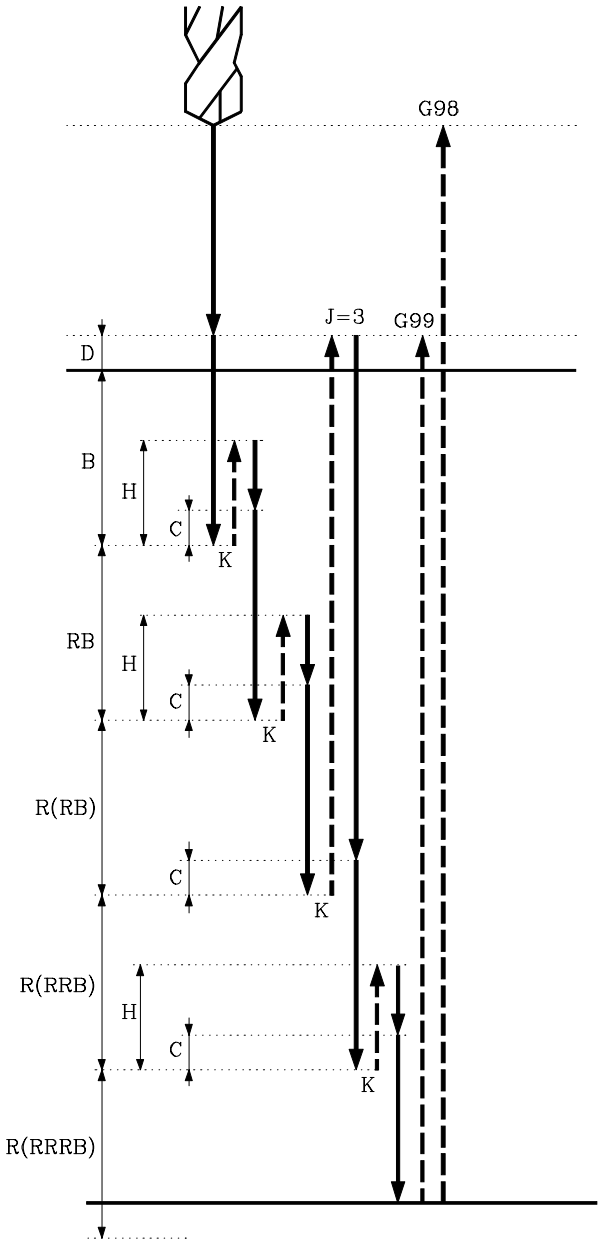
If R is not equal to 1, the first drilling step will be "B", the second, "R B", the third "R (RB)", and so on, i.e., after the second step, the new step will be the product of factor R by the previous step.

If R is selected with a value other than 1, the CNC will not allow smaller steps than that programmed in L.

Basic operation:

1. If the spindle was in operation previously, its turning direction is maintained. If it was not in movement, it will start by turning clockwise (M03).

2. Rapid movement of the longitudinal axis from the initial plane to the reference plane.



3. First drilling operation. Movement at working feedrate of the longitudinal axis to the programmed incremental depth in “B+D”.

   This movement will be carried out either in G07 or G50 depending on the value assigned to the longitudinal axis "INPOSW2(P51)"

   If P51 =0, in G7 (square corner) If P51=1, in G50 (controlled round corner).

4. Drilling loop. The following steps will be repeated until the machining depth coordinate programmed in I is reached.

   4.1. Dwell K in hundredths of a second, if this has been programmed.

   4.2. Withdrawal of the longitudinal axis in rapid (G00) as far as the reference plane, if the number of drillings programmed in J were made, otherwise it withdraws the distance programmed in “H”.

   4.3. Longitudinal axis approach in rapid (G00) as far as a distance “C” of the previous drilling step.

   4.4. Another drilling step. Movement of the longitudinal axis, at the working feedrate (G01) until the next incremental drilling according to “B and R”.

   This movement will be carried out in either in G07 or in G50 depending on the value assigned to the parameter of the longitudinal axis "INPOSW2(P51)".

   If P51=0 in G7 (square corner). If P51=1, in G50 (controlled round corner).

5. Dwell time K in hundredths of a second, if this has been programmed.

6. Withdrawal at rapid feedrate (G00) of the longitudinal axis to the initial or reference plane, depending on whether G98 or G99 has been programmed.

If a scaling factor is applied to this cycle, it should be borne in mind that this scaling factor will only affect the reference plane coordinates and drilling depth.

Therefore, and due to the fact that parameter “D” is not affected by the scaling factor, the surface coordinate of the part will not be proportional to the programmed cycle.

Programming example supposing that the work plane is formed by the X and Y axes, that the longitudinal axis is the Z axis and that the starting point is X0 Y0 Z0:

```
T1
M6
G0 G90 X0 Y0 Z0 ..... ; Starting point
G69 G98 G91 X100 Y25 Z-98 I-52 B12 C2 D2
          H5 J2 K150 L3 R0.8 F100 S500 M8 ..... ; Canned cycle definition
G80 ..... ; Canned cycle cancellation
G90 X0 Y0 ..... ; Positioning
M30 ..... ; End of program
```

9.5.2. G81 DRILLING CANNED CYCLE

This cycle drills at the point indicated until the final programmed coordinate is reached.

It is possible to program a dwell at the bottom of the drill hole.

Working in cartesian coordinates, the basic structure of the block is as follows:

G81 G98/G99 X Y Z I K

G98 The tool withdraws to the Initial Plane, once the hole has been drilled.

G99 The tool withdraws to the Reference Plane, once the hole has been drilled.

XY±5.5 These are optional and define the movement of the axes of the main plane to position the tool at the machining point.

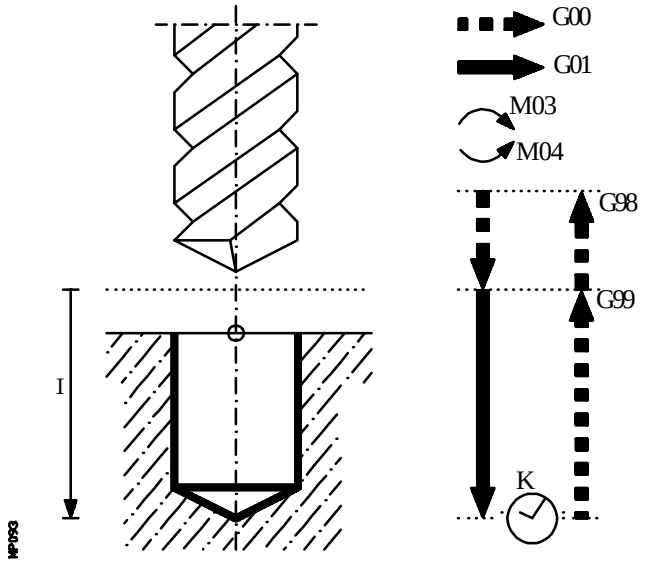
This point can be programmed in cartesian coordinates or in polar coordinates, and the coordinates may be absolute or incremental, according to whether the machine is operating in G90 or G91.

Z±5.5 Defines the reference plane coordinate. It can be programmed in absolute coordinates or incremental coordinates and in this case will be referred to the initial plane.

If this is not programmed, the CNC will take the position occupied by the tool at that moment as the reference plane.

I±5.5 Defines drilling depth. It can be programmed in absolute coordinates or incremental coordinates and in this case will be referred to the reference plane.

K5 Defines the dwell time, in hundredths of a second, after each drilling step, until the withdrawal begins. Should this not be programmed, the CNC will take a value of K0.



Basic operation:

1. If the spindle was in operation previously, its turning direction is maintained. If it was not in movement, it will start by turning clockwise (M03).

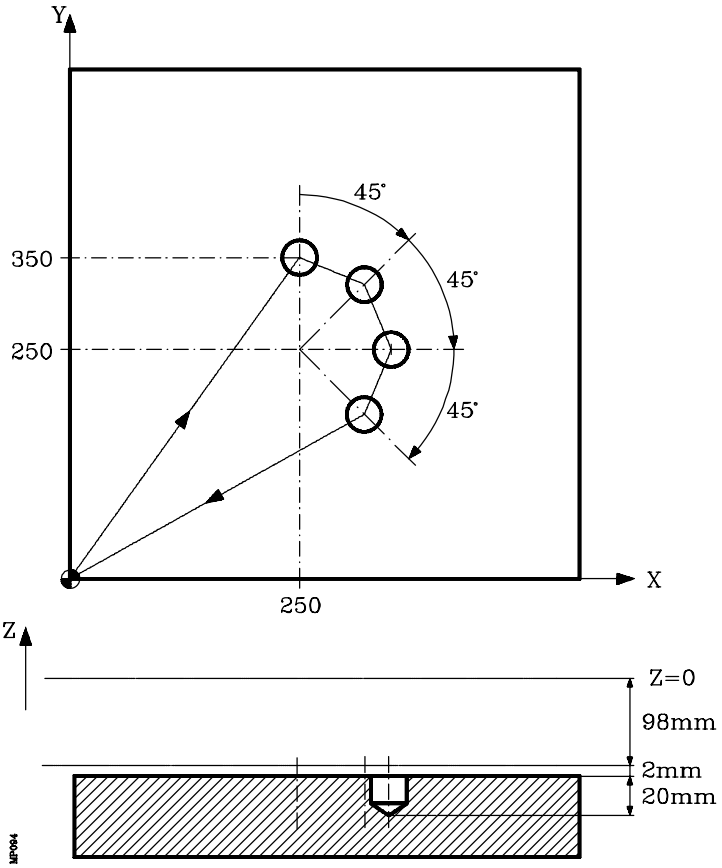
2. Rapid movement of the longitudinal axis from the initial plane to the reference plane.

3. The hole is drilled. Movement at working feedrate of the longitudinal axis to the programmed machining depth I.

4. Dwell time K in hundredths of a second, if this has been programmed.

5. Withdrawal at rapid feedrate (G00) of the longitudinal axis to the initial or reference plane, depending on whether G98 or G99 has been programmed.

Programming example assuming that the work plane is formed by the X and Y axes, that the longitudinal axis is the Z axis and that the starting point is X0 Y0 Z0:




```
T1
M6
G0 G90 X0 Y0 Z0 ..... ; Starting point
G81  G98  G00 G91 X250 Y350 Z-98 I-22 F100 S500 ..... ; Positioning and definition of canned cycle
G93  I250 J250 ..... ; Sets polar coordinate origin
Q-45 N3 ..... ; Turn and canned cycle, 3 times
G80 ..... ; Cancels canned cycle
G90 X0 Y0 ..... ;  Positioning
M30 ..... ; End of program
```

9.5.3. G82. DRILLING CANNED CYCLE WITH DWELL

This cycle drills at the point indicated until the final programmed coordinate is reached.

Then it executes a dwell at the bottom of the drill hole.

Working in cartesian coordinates, the basic structure of the block is as follows:

G82 G98/G99 X Y Z I K

G98 The tool withdraws to the Initial Plane, once the hole has been drilled.

G99 The tool withdraws to the Reference Plane, once the hole has been drilled.

XY±5.5 These are optional and define the movement of the axes of the main plane to position the tool at the machining point.

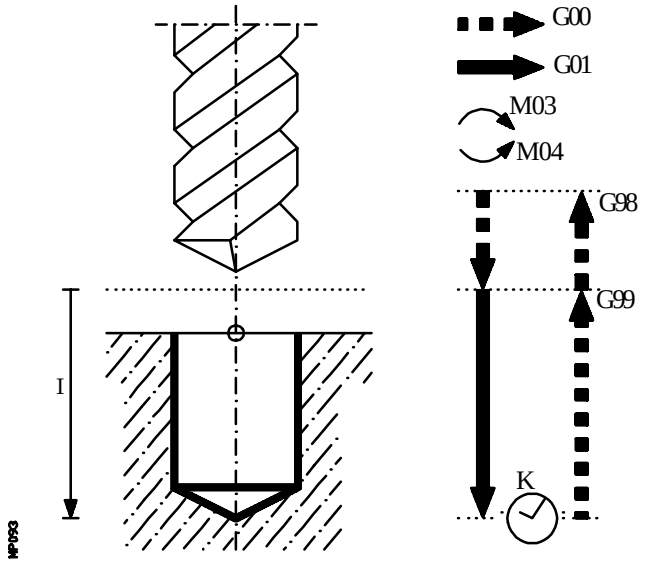
This point can be programmed in cartesian coordinates or in polar coordinates, and the coordinates may be absolute or incremental, according to whether the machine is operating in G90 or G91.

Z±5.5 Defines the reference plane coordinate. It can be programmed in absolute coordinates or incremental coordinates and in this case will be referred to the initial plane.

If this is not programmed, the CNC will take the position occupied by the tool at that moment as the reference plane.

I±5.5 Defines drilling depth. It can be programmed in absolute coordinates or incremental coordinates and in this case will be referred to the reference plane.

K5 Defines the dwell time, in hundredths of a second, after each drilling step, until the withdrawal begins. Should this not be programmed, the CNC will take a value of K0.



Basic operation:

1. If the spindle was in operation previously, its turning direction is maintained. If it was not in movement, it will start by turning clockwise (M03).

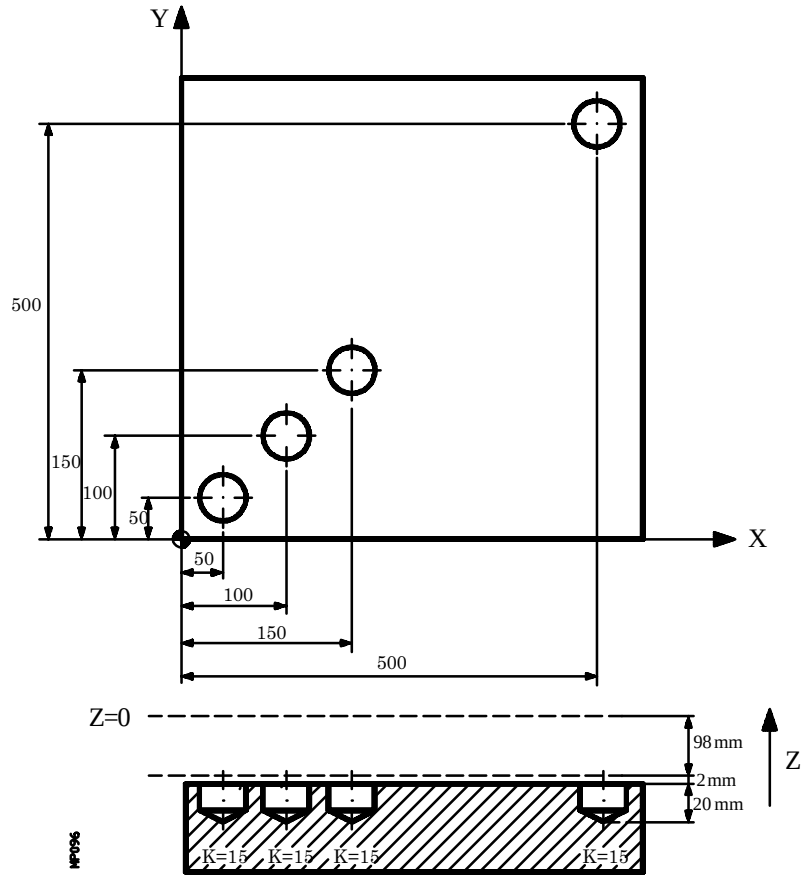
2. Rapid movement of the longitudinal axis from the initial plane to the reference plane.

3. The hole is drilled. Movement at working feedrate of the longitudinal axis to the bottom of the machined hole, programmed in I.

4. Dwell time K in hundredths of a second.

5. Withdrawal at rapid feedrate (G00) of the longitudinal axis to the initial or reference plane, according to whether G98 or G99 has been programmed.

Programming example assuming that the work plane is formed by the X and Y axes, that the longitudinal axis is the Z axis and that the starting point is X0 Y0 Z0:




```
T1
M6
G0 G90 X0 Y0 Z0 ..... ; Starting point
G82 G99 G00 G91 X50 Y50 Z-98 I-22 K150 F100 S500 N3 .... ; 3 machining positions
G98 G90 G00 X500 Y500 ..... ;  Positioning and canned cycle
G80 ..... ;  Cancels canned cycle
G90 X0 Y0 ..... ; Positioning
M30 ..... ; End of program
```

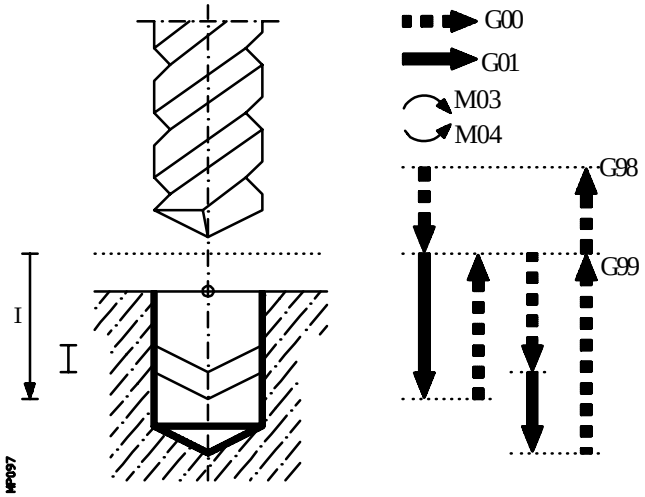
9.5.4. G83. SIMPLE DEEP HOLE DRILLING

This cycle performs successive drilling steps until the final programmed coordinate is reached.

The tool withdraws as far as the reference plane after each drilling step.

Working in cartesian coordinates, the basic structure of the block is as follows:

G83 G98/G99 X Y Z I J



G98 The tool withdraws to the Initial Plane, once the hole has been drilled.

G99 The tool withdraws to the Reference Plane, once the hole has been drilled.

XY±5.5 These are optional and define the movement of the axes of the main plane to position the tool at the machining point.

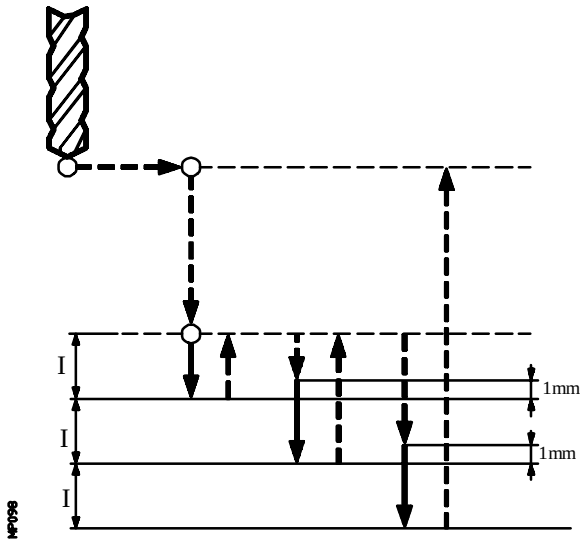
This point can be programmed in cartesian coordinates or in polar coordinates, and the coordinates may be absolute or incremental, according to whether the machine is operating in G90 or G91.

Z±5.5 Defines the reference plane coordinate. It can be programmed in absolute coordinates or incremental coordinates and in this case will be referred to the initial plane.

If this is not programmed, the CNC will take the position occupied by the tool at that moment as the reference plane.

I±5.5 Defines the value of each drilling step according to the axis longitudinal to the main plane.

J4 Defines the number of steps which the drill is to make. This can be programmed with a value between 1 and 9999.



Basic operation

1. If the spindle was in operation previously, its turning direction is maintained. If it was not in movement, it will start by turning clockwise (M03).

2. Rapid movement of the longitudinal axis from the initial plane to the reference plane.

3. First drilling. Movement at working feedrate of the longitudinal axis to the programmed incremental depth in “I”.

   This movement will be carried out either in G07 or G50 depending on the value assigned to the longitudinal axis "INPOSW2(P51)"

   If P51 =0, in G7 (square corner) otherwise, in G50 (controlled round corner).

4. Drilling loop. The following steps will be repeated “J-1” times as in the previous step the first programmed drilling was done.

   4.1. Withdrawal of the longitudinal axis in rapid (G00) to the reference plane.

   4.2. Longitudinal axis approach in rapid (G00):

   If INPOSW2=0 up to 1mm from the previous drilling peck.
   Otherwise, up to "INPOSW2 +0.02 MM of the previous drilling peck.

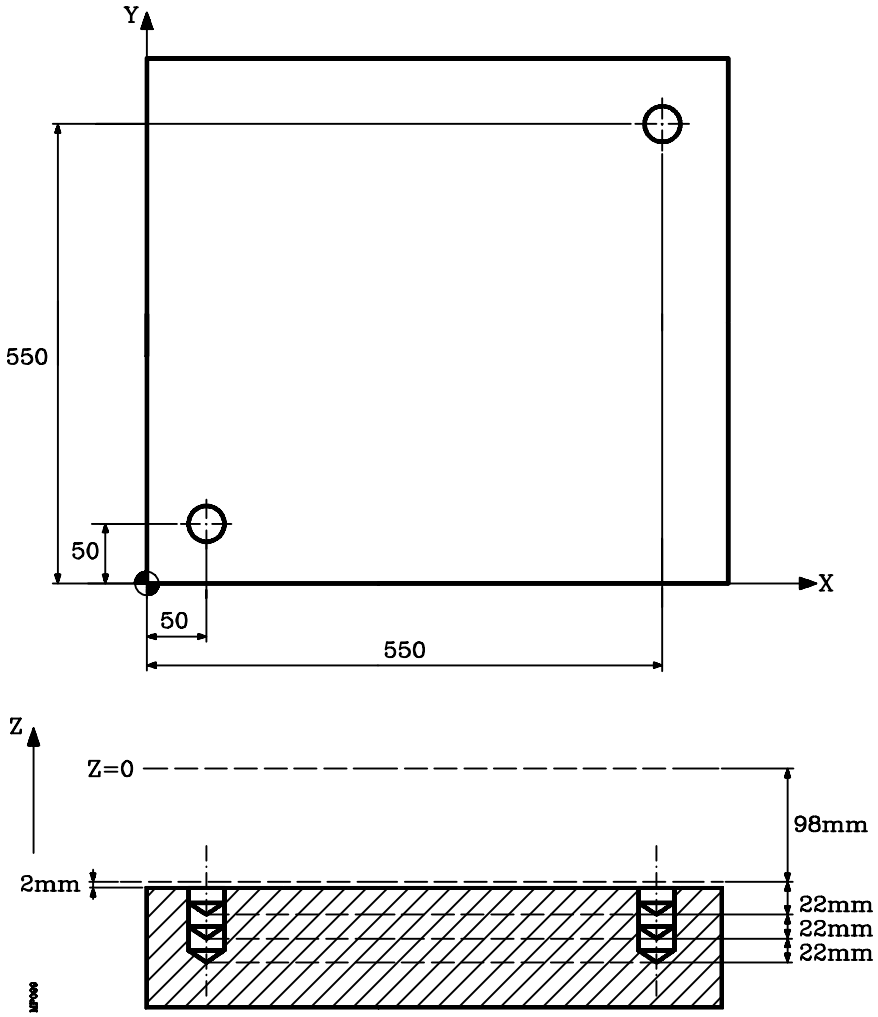
   4.3. Another drilling step. Movement of the longitudinal axis, at working feedrate (G01) the incremental depth programmed in “I”.

   If INPOSW2= 0 in G7 Otherwise, in G50

5. Withdrawal at rapid feedrate (G00) of the longitudinal axis to the initial or reference plane, depending on whether G98 or G99 has been programmed.

If a scaling factor is applied to this cycle, drilling will be performed proportional to that programmed, with the same step “I” programmed, but varying the number of steps “J”.

Programming example assuming that the work plane is formed by the X and Y axes, that the longitudinal axis is the Z axis and that the starting point is X0 Y0 Z0:




```
T1
M6
G0 G90 X0 Y0 Z0 ..... ; Starting point
G83 G99 G00 G90 X50 Y50 Z-98 I-22 J3 F100 S500 M4 ..... ; Positioning and canned cycle setting
G98 G00 G91 X500 Y500 ..... ; Positioning and canned cycle
G80 ..... ; Cancels canned cycle
G90 X0 Y0 ..... ; Positioning
M30 ..... ; End of program
```

9.5.5. G84. TAPPING CANNED CYCLE

This cycle taps at the point indicated until the final programmed coordinate is reached. The general logic output "**TAPPING**" (M5517) will stay active during this cycle.

Due to the fact that the tapping tool turns in two directions (one when tapping and the other when withdrawing from the thread), by means of the machine parameter of the spindle "SREVM05" it is possible to select whether the change in turning direction is made with the intermediate spindle stop, or directly.

General machine parameter "STOPAP(P116)" indicates whether general inputs /STOP, /FEEDHOL and /XFERINH are enabled or not while executing function G84.

It is possible to program a dwell before each reversal of the spindle turning direction, i.e., at the bottom of the thread hole and when returning to the reference plane.

Working in cartesian coordinates, the basic structure of the block is as follows:

G84 G98/G99 X Y Z I K R

G98 The tool withdraws to the Initial Plane, once the hole has been tapped.

G99 The tool withdraws to the Reference Plane, once the hole has been tapped.

XY±5.5 These are optional and define the movement of the axes of the main plane to position the tool at the machining point.

This point can be programmed in cartesian coordinates or in polar coordinates, and the coordinates may be absolute or incremental, according to whether the machine is operating in G90 or G91.

Z±5.5 Defines the reference plane coordinate. It can be programmed in absolute coordinates or incremental coordinates and in this case will be referred to the initial plane.

If this is not programmed, the CNC will take the position occupied by the tool at that moment as the reference plane.

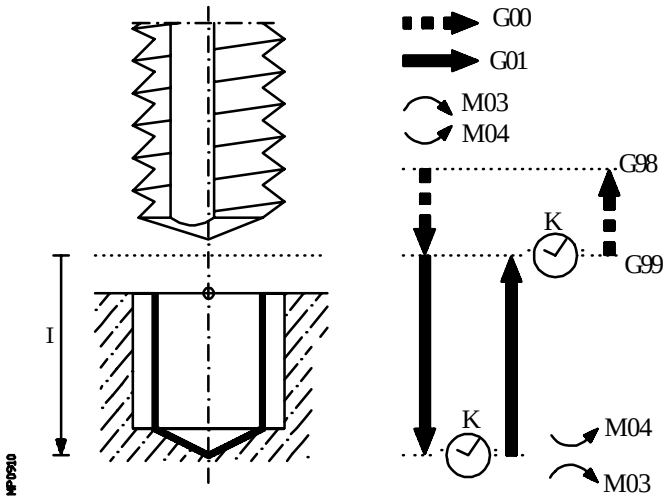
I±5.5 Defines tapping depth. It can be programmed in absolute coordinates or incremental coordinates and in this case will be referred to the reference plane.

K5 Defines the dwell time, in hundredths of a second, after each drilling step, until the withdrawal begins. Should this not be programmed, the CNC will take a value of K0.

R Defines the type of tapping cycle to be performEd: normal if "R0" and rigid if "R1".

To perform a rigid tapping cycle, the spindle must be installed so it can work in closed loop; i.e. with encoder and servo drive.

During rigid tapping the CNC interpolates the longitudinal axis with the spindle rotation.



Basic operation

1. If the spindle was in operation previously, its turning direction is maintained. If it was not in movement, it will start by turning clockwise (M03).

2. Rapid movement of the longitudinal axis from the initial plane to the reference plane.

3. Movement of the longitudinal axis and at the working feedrate, to the bottom of the machined section, producing the threaded hole. The canned cycle will execute this movement and all later movements at 100% of F feedrate and the programmed S speed.

   If rigid tapping is selected (parameter R=1), the CNC will activate the general logic output “RIGID” (M5521) to indicate to the PLC that a rigid tapping block is being executed.

4. Spindle stop (M05). This will only be performed when the spindle meachine parameter **“SREVM05”** is selected and parameter "K" has a value other than "0"..

5. Dwell, if parameter “K” has been programmed.

6. Spindle turning direction reversal.

7. Withdrawal, at working feedrate, of the longitudinal axis as far as the reference plane. Once this coordinate has been reached, the canned cycle will assume the selected FEEDRATE OVERRIDE and the SPINDLE OVERRIDE.

   If rigid tapping is selected (parameter R=1), the CNC will activate the general logic output “RIGID” (M5521) to indicate to the PLC that a rigid tapping block is being executed.

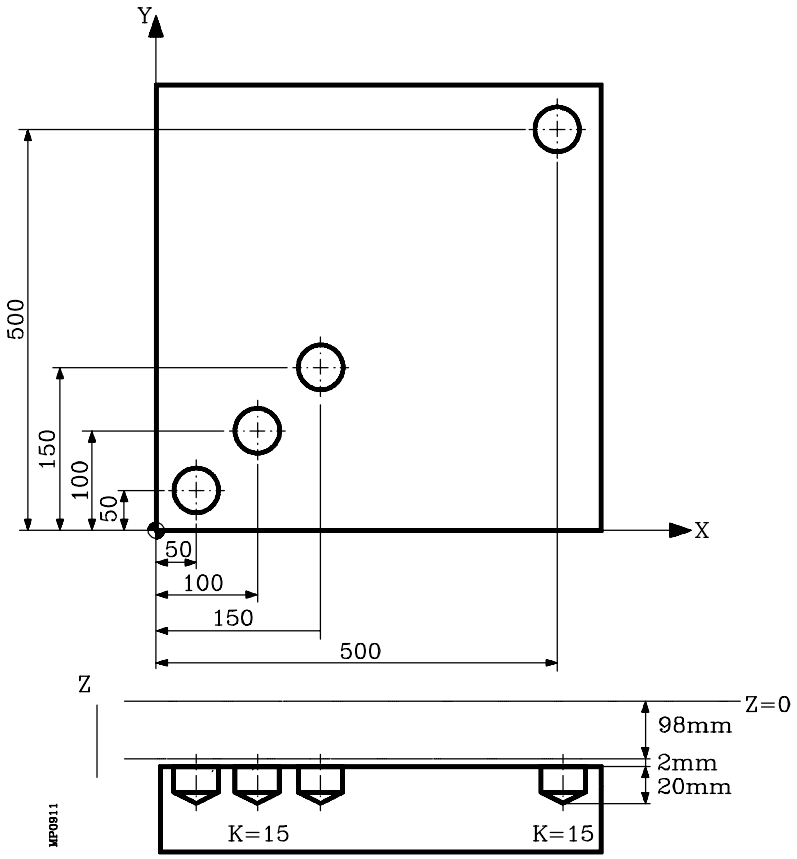
8. Spindle stop (M05). This will only be performed if the spindle meachine parameter **“SREVM05”** is selected.

9. Dwell, if parameter “K” has been programmed.

10. Spindle turning direction reversal.

11. Withdrawal, at rapid feedrate (G00), of the longitudinal axis as far as the initial plane if G98 has been programmed.

Programming example assuming that the work plane is formed by the X and Y axes, that the longitudinal axis is the Z axis and that the starting point is X0 Y0 Z0:




```
T1
M6
G0 G90 X0 Y0 Z0 ..... ; Starting point
G84 G99 G00 G91 X50 Y50 Z-98 I-22 K150 F350 S500 N3 ..... ; 3 machining positions
G98 G00 G90 X500 Y500 ..... ; Positioning and canned cycle
G80 ..... ; Cancels canned cycle
G90 X0 Y0 ..... ; Positioning
M30 ..... ; End of program
```

9.5.6. G85. REAMING CYCLE

This cycle reams at the point indicated until the final programmed coordinate is reached.

It is possible to program a dwell at the bottom of the machined hole.

Working in cartesian coordinates, the basic structure of the block is as follows:

G85 G98/G99 X Y Z I K

G98 The tool withdraws to the Initial Plane, once the hole has been reamed**.**

G99 The tool withdraws to the Reference Plane, once the hole has been reamed.

XY±5.5 These are optional and define the movement of the axes of the main plane to position the tool at the machining point.

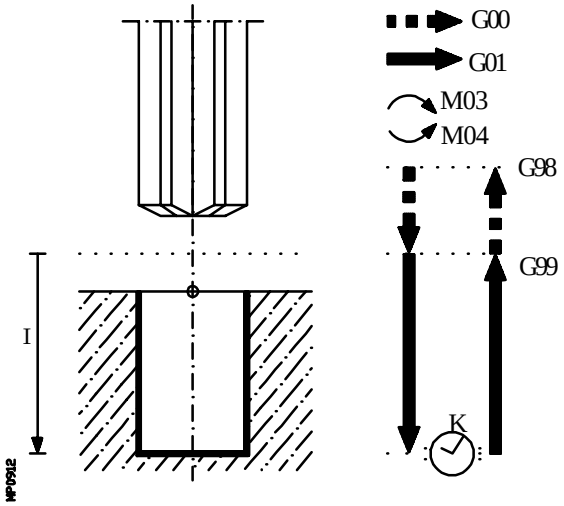
This point can be programmed in cartesian coordinates or in polar coordinates, and the coordinates may be absolute or incremental, according to whether the machine is operating in G90 or G91**.**

Z±5.5 Defines the reference plane coordinate. It can be programmed in absolute coordinates or incremental coordinates and in this case will be referred to the initial plane.

If this is not programmed, the CNC will take the position occupied by the tool at that moment as the reference plane**.**

I±5.5 Defines reaming depth. It can be programmed in absolute coordinates or incremental coordinates and in this case will be referred to the reference plane.

K5 Defines the dwell time, in hundredths of a second, after each drilling step, until the withdrawal begins. Should this not be programmed, the CNC will take a value of K0.



Basic operation

1. If the spindle was in operation previously, its turning direction is maintained. If it was not in movement, it will start by turning clockwise (M03).

2. Rapid movement of the longitudinal axis from the initial plane to the reference plane.

3. Movement at the working feedrate (G01) of the longitudinal axis to the bottom of the machined hole, and reaming.

4. Dwell, if parameter "K" has been programmed.

5. Withdrawal at working feedrate, of the longitudinal axis as far as the reference plane.

6. Withdrawal, at rapid feedrate (G00), of the longitudinal axis as far as the initial plane if G98 has been programmed.

Programming example assuming that the work plane is formed by the X and Y axes, that the longitudinal axis is the Z axis and that the starting point is X0 Y0 Z0:

```
T1
M6
G0 G90 X0 Y0 Z0 ..... ;Starting point
G85 G98 G91 X250 Y350 Z-98 I-22 F100 S500 ..... ;Canned cycle definition
G80 ..... ;Canned cycle cancellation
G90 X0 Y0 ..... ;Positioning
M30 ..... ;End of program
```

9.5.7. G86. BORING CYCLE WITH WITHDRAWAL IN RAPID (G00)

This cycle bores at the point indicated until the final programmed coordinate is reached.

It is possible to program a dwell at the bottom of the machined hole.

Working in cartesian coordinates, the basic structure of the block is as follows:

G86 G98/G99 X Y Z I K

G98 The tool withdraws to the Initial Plane, once the hole has been bored.

G99 The tool withdraws to the Reference Plane, once the hole has been bored.

XY±5.5 These are optional and define the movement of the axes of the main plane to position the tool at the machining point.

This point can be programmed in cartesian coordinates or in polar coordinates, and the coordinates may be absolute or incremental, according to whether the machine is operating in G90 or G91.

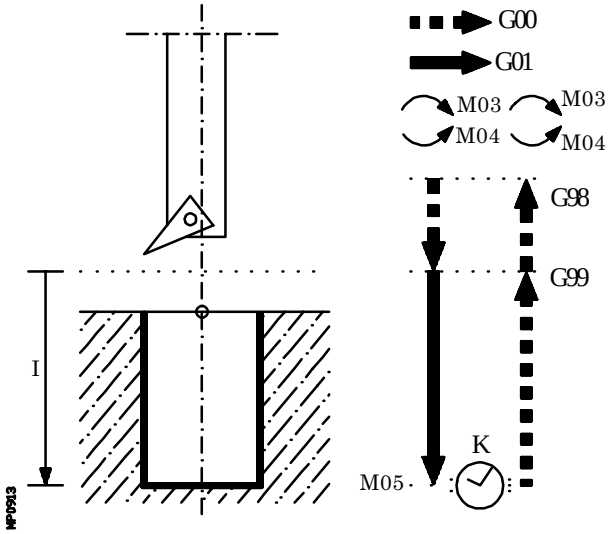
Z±5.5 Defines the reference plane coordinate. It can be programmed in absolute coordinates or incremental coordinates and in this case will be referred to the initial plane.

If this is not programmed, the CNC will take the position occupied by the tool at that moment as the reference plane.

I±5.5 Defines boring depth. It can be programmed in absolute coordinates or incremental coordinates and in this case will be referred to the reference plane.

K5 Defines the dwell time, in hundredths of a second, after each drilling step, until the withdrawal begins. Should this not be programmed, the CNC will take a value of K0.

Basic operation



1. If the spindle was in operation previously, its turning direction is maintained. If it was not in movement, it will start by turning clockwise (M03).

2. Rapid movement of the longitudinal axis from the initial plane to the reference plane.

3. Movement at the working feedrate (G01) of the longitudinal axis to the bottom of the machined hole, and boring.

4. Spindle stop (M05).

5. Dwell, if parameter "K" has been programmed.

6. Withdrawal, at rapid feedrate (G00), of the longitudinal axis as far as the initial plane or the reference plane, depending on whether G98 or G99 has been programmed.

7. When spindle withdrawal has been completed, it will start in the same direction in which it was turning before.

Programming example assuming that the work plane is formed by the X and Y axes, that the longitudinal axis is the Z axis and that the starting point is X0 Y0 Z0:

```
T1
M6
G0 G90 X0 Y0 Z0 ..... ;Starting point
G86 G98 G91 X250 Y350 Z-98 I-22 K20 F100 S500 .. ;Canned cycle definition
G80 ..... ;Canned cycle cancellation
G90 X0 Y0 ..... ;Positioning
M30 ..... ;End of program
```

9.5.8. G87. RECTANGULAR POCKET CANNED CYCLE

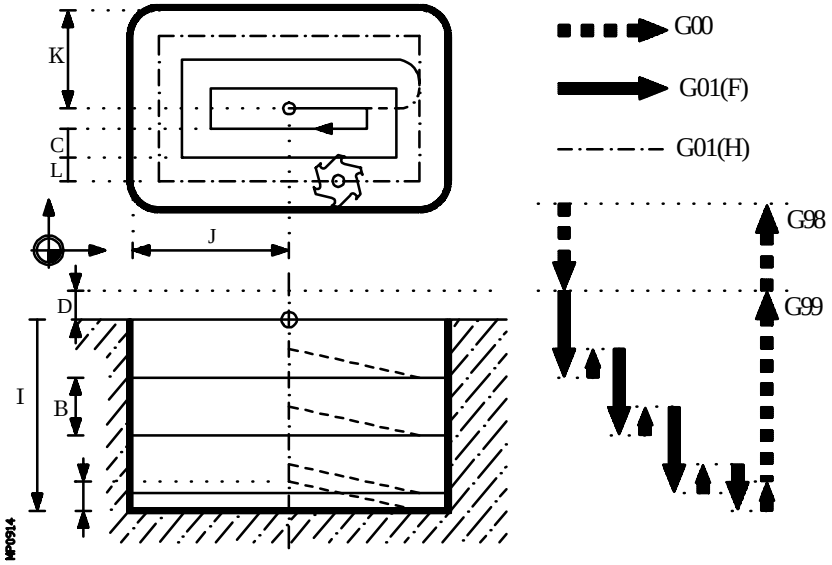
This cycle executes a rectangular pocket at the point indicated until the final programmed coordinate is reached.

It is possible to program, in addition to milling pass and feedrate, a final finishing step with its corresponding milling feedrate.

In order to obtain a good finish in the machining of the pocket walls, the CNC will apply a tangential entry and exit to the last milling step during each cutting operation.

Working in cartesian coordinates, the basic structure of the block is as follows:

G87 G98/G99 X Y Z I J K B C D H L V



G98 The tool withdraws to the Initial Plane, once the pocket has been made.

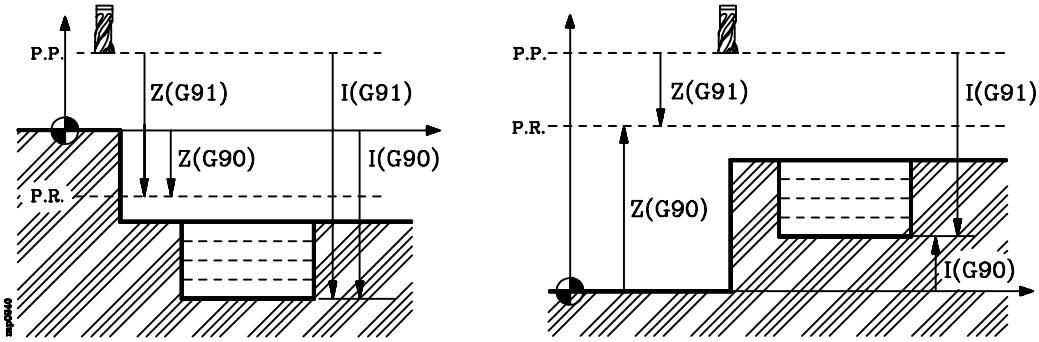
G99 The tool withdraws to the Reference Plane, once the pocket has been made.

XY±5.5 These are optional and define the movement of the axes of the main plane to position the tool at the machining point.

This point can be programmed in cartesian coordinates or in polar coordinates, and the coordinates may be absolute or incremental, according to whether the machine is operating in G90 or G91.

Z±5.5 Defines the reference plane coordinate.

When programmed in absolute coordinates, it will be referred to the part zero and when programmed in incremental coordinates, it will be referred to the starting plane (P.P.).

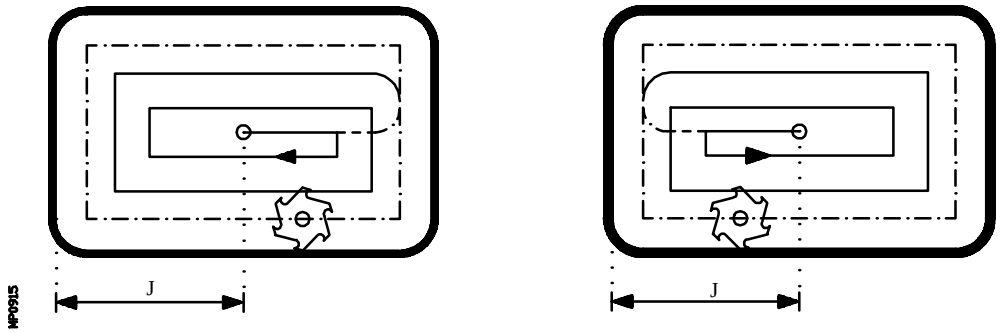


If this is not programmed, the CNC will take the position occupied by the tool at that moment as the reference plane. Thus, the starting plane (P.P.) and the reference plane (P.R.) wil be the same.

I±5.5 Defines machining depth.

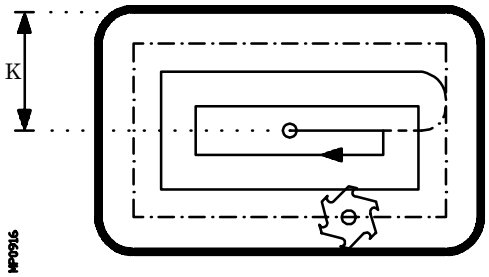
When programmed in absolute coordinates, it will be referred to the part zero and when programmed in incremental coordinates, it will be referred to the starting plane (P.P.).

J±5.5 Defines the distance from the center to the edge of the pocket according to the abscissa axis. The sign indicates the pocket machining direction.



J with "+" sign J with "-" sign

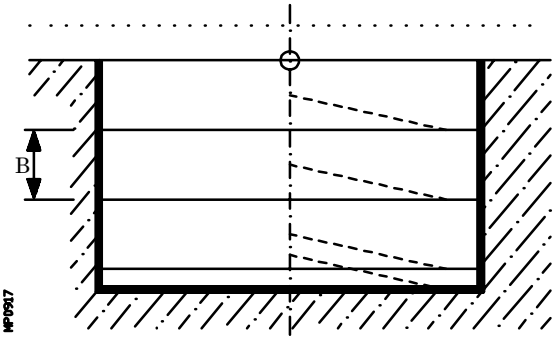
K5.5 Defines the distance from the center to the edge of the pocket according to the ordinate axis.



B±5.5 Defines the cutting depth according to the longitudinal axis.

- If this is programmed with a positive sign, the entire cycle will be executed with the same machining pass, this being equal to or less than that programmed.

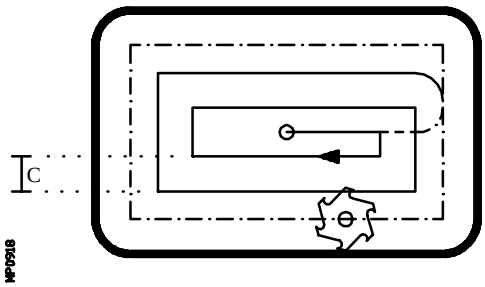
- If this is programmed with a negative sign, the entire pocket will be executed with the given pass, except for the last pass which will machine the rest.



C±5.5 Defines the milling pass along the main plane.

- If the value is positive, the entire cycle will be executed with the same milling step, this being equal to or less than that programmed.

- If the value is negative, the entire pocket will be executed with the given step, except for the last step which will machine whatever remains.



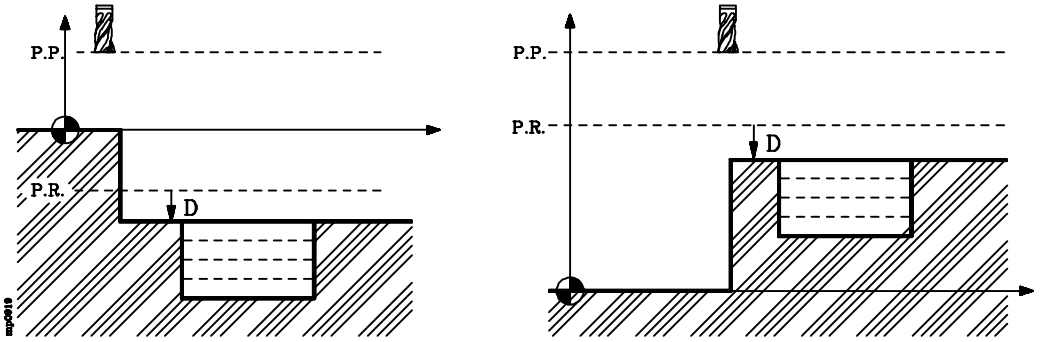
If this is not programmed, the CNC will assume 3/4 of the diameter of the diameter of the selected tool.

If programmed with a value greater than the tool diameter, the CNC will issue the corresponding error.

If programmed with a value of 0, the CNC will show the corresponding error.

D5.5 Defines the distance between the reference plane and the surface of the part where the pocket is to be made.

During the first deepening operation this amount will be added to incremental depth “B”. If this is not programmed, a value of 0 will be taken.

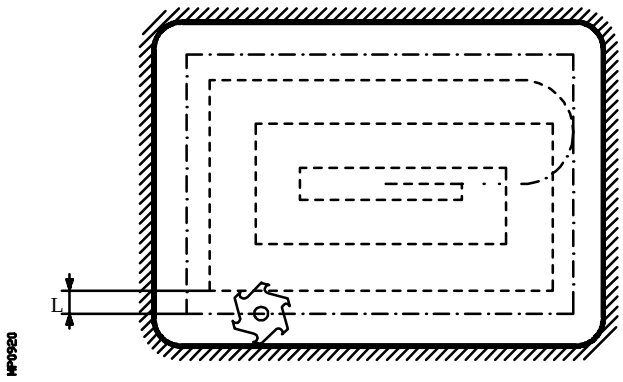


H.5.5 Defines the working feedrate during the finishing pass.

If this is not programmed or is programmed with a value of 0, the value of the working feedrate for machining will be taken.

L±5.5 Defines the value of the finishing pass, along the main plane.

- If the value is positive, the finishing pass is made on a square corner (G07).
- If the value is negative, the finishing pass is made on a rounded corner (G05).



If this is not programmed or is programmed with a value of 0 no finishing pass will be made.

V.5.5 Defines the tool penetrating feedrate.

If not programmed or programmed with a value of "0", the CNC will assume 50% of the feedrate in the plane (F).

Basic operation

1. If the spindle was in operation previously, its turning direction is maintained. If it was not in movement, it will start by turning clockwise (M03).

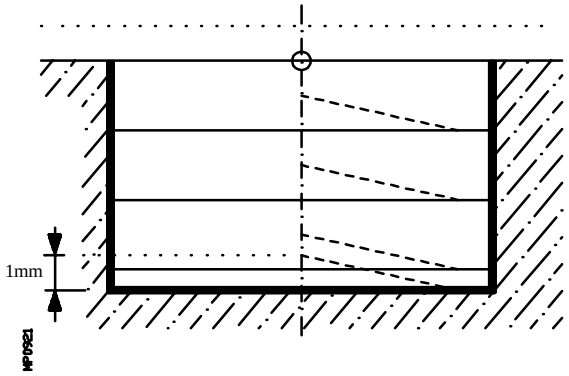
2. Rapid movement of the longitudinal axis from the initial plane to the reference plane.

3. First deepening operation. Movement of longitudinal axis at the feedrate indicated by "V" to the incremental depth programmed in “B+D”.

4. Milling at the working feedrate of the surface of the pocket in steps defined by means of “C” as far as a distance “L” (finishing pass) from the pocket wall.

5. Milling of the “L” finishing pass with the working feedrate defined in “H”.

6. Once the finishing pass has been completed, the tool withdraws at the rapid feedrate (G00) to the center of the pocket, the longitudinal axis being separated 1 mm (0.040 inch) from the machined surface.

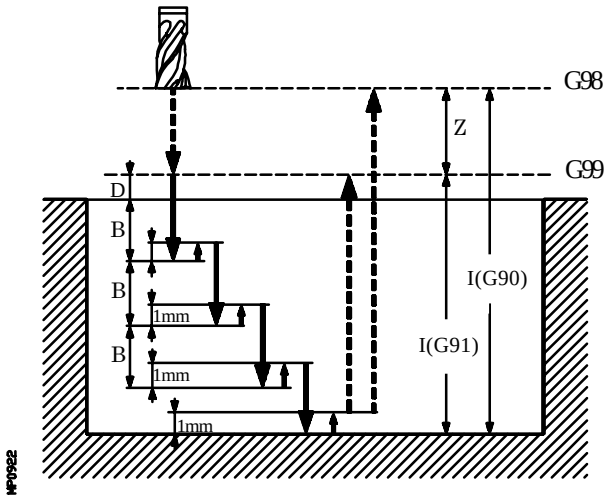


7. Further milling runs until the total depth of the pocket is reached.

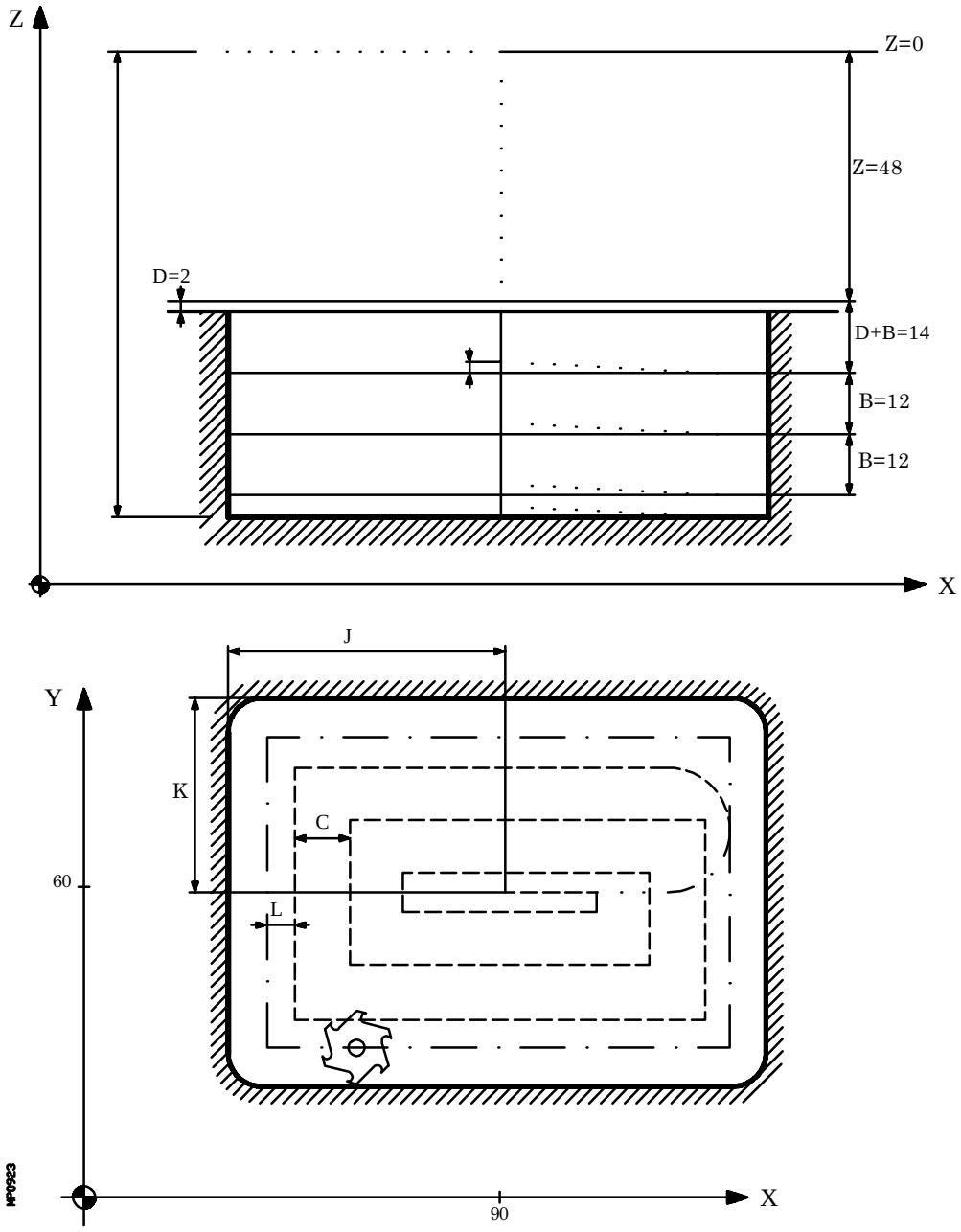
   - Movement of the longitudinal axis at the feedrate indicated by "V", up to a distance “B” from the previous surface.

   - Milling of a new surface following the steps indicated in paragraphs 4, 5 and 6.

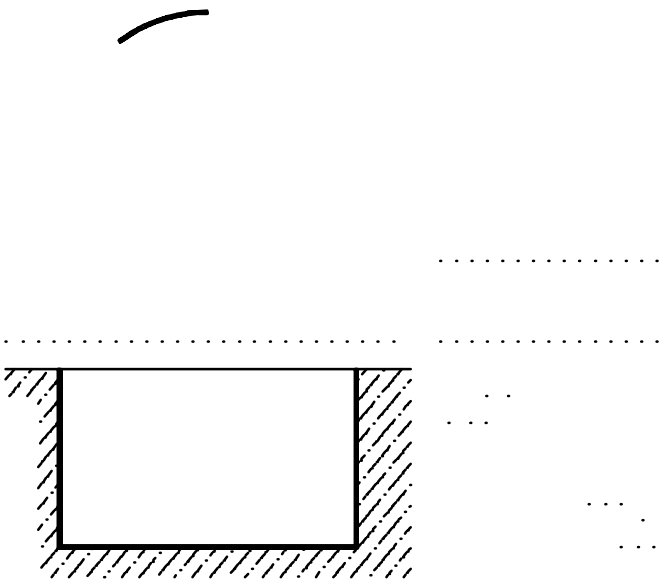
8. Withdrawal at rapid feedrate (G00) of the longitudinal axis to the initial or reference plane, along depending on G98 or G99 has been programmed.



Programming example assuming that the work plane is formed by the X and Y axes, that the longitudinal axis is the Z axis and that the starting point is X0 Y0 Z0:




```
(TOR1=6, TOT1=0)
T1 D1
M6
G0 G90 X0 Y0 Z0 ..... ; Starting point
G87 G98 G00 G90 X90 Y60 Z-48 I-90 J52.5 K37.5 B12
        C10 D2 H100 L5 V100 F300 S1000 T1 D1 M03 ..... ; Canned cycle definition
G80 ..... ;Cancels canned cycle
G90 X0 Y0 ..... ; Positioning
M30 ..... ; End of program
```

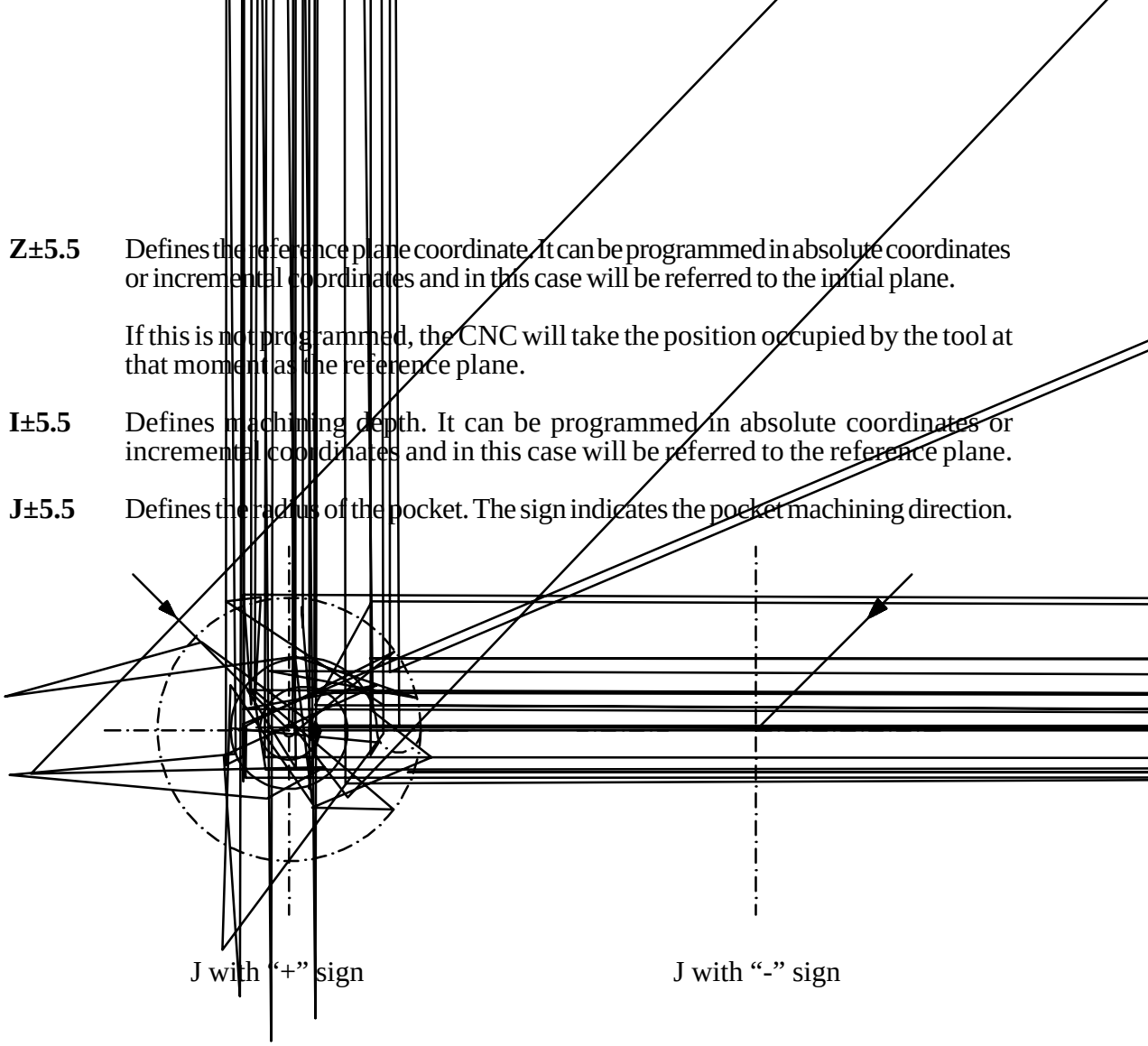


Z±5.5 Defines the reference plane coordinate. It can be programmed in absolute coordinates or incremental coordinates and in this case will be referred to the initial plane.

If this is not programmed, the CNC will take the position occupied by the tool at that moment as the reference plane.

I±5.5 Defines machining depth. It can be programmed in absolute coordinates or incremental coordinates and in this case will be referred to the reference plane.

J±5.5 Defines the radius of the pocket. The sign indicates the pocket machining direction.



J with "+" sign J with "-" sign

B±5.5 Defines the cutting pass along the longitudinal axis to the main plane.

- If this value is positive, the entire cycle will be executed with the same machining pass, this being equal to or less than that programmed.

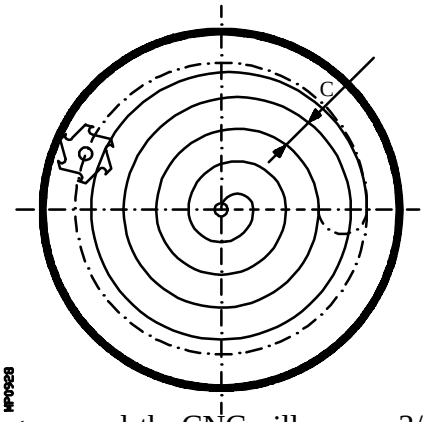
- If this value is negative, the entire pocket will be executed with the given pass, except for the last pass which will machine the rest.

C±5.5 Defines the milling pass along the main plane.

- If the value is positive, the entire cycle will be executed with the same millingJ

pass, this being equal to or less than that programmed.

- If the value is negative, the entire pocket will be executed with the given pass, except for the last pass which will machine whatever remains.



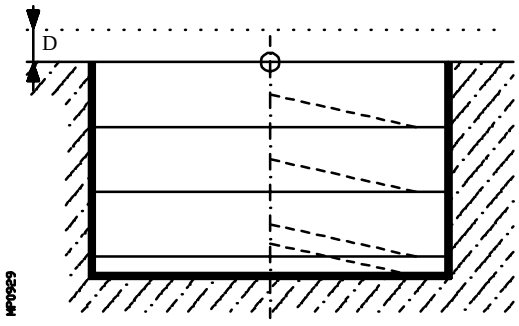
If this is not programmed, the CNC will assume 3/4 of the diameter of the diameter of the selected tool.

If programmed with a value greater than the tool diameter, the CNC will issue the corresponding error.

If programmed with a value of 0, the CNC will show the corresponding error.

D5.5 Defines the distance between the reference plane and the surface of the part where the pocket is to be made.

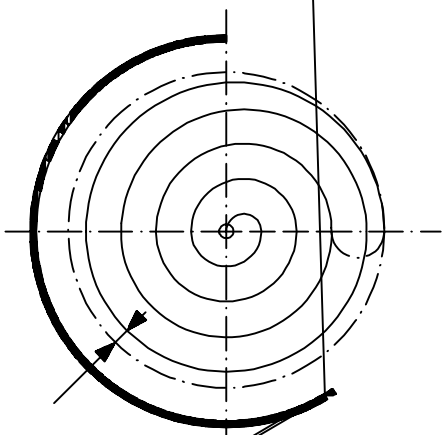
During the first deepening operation this amount will be added to incremental depth "B". If this is not programmed, a value of 0 will be taken.



H.5 Defines the working feedrate during the finishing pass.

If this is not programmed or is programmed with a value of 0, the value of the working feedrate for machining will be taken.

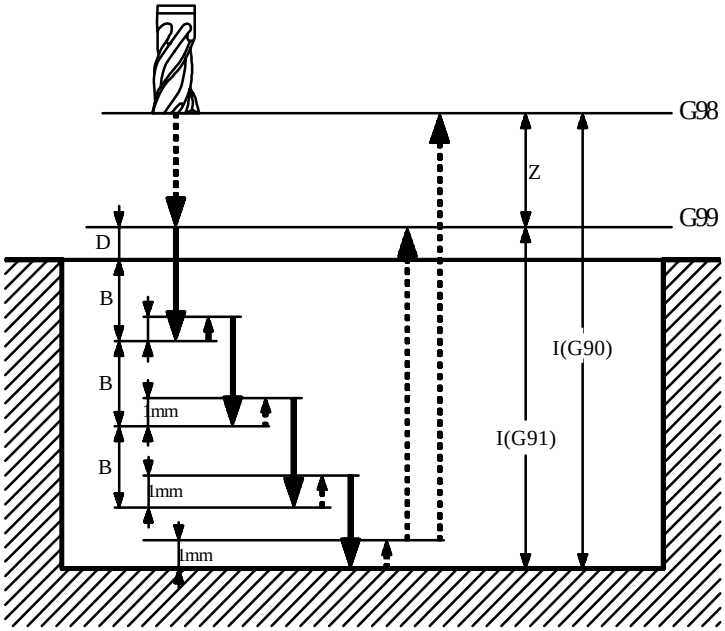
L5.5 Defines the value of the finishing pass, along the main plane.

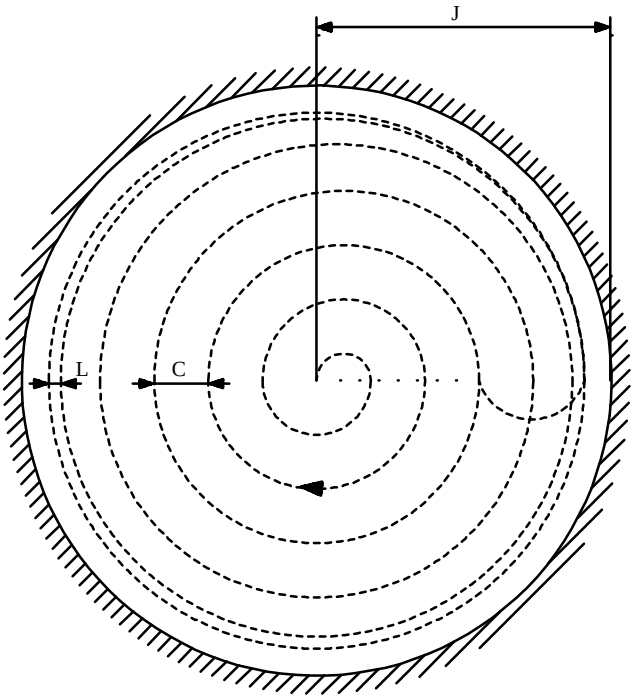


If this is not programmed or is programmed with a value of 0 no finishing pass will be made.

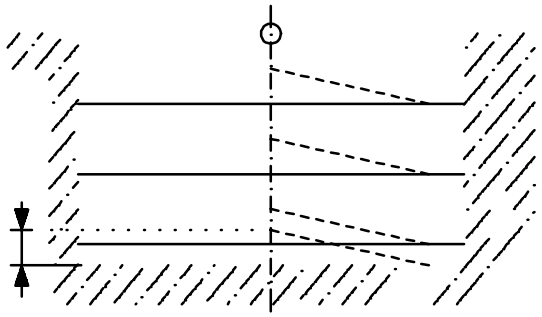
V.5.5 Defines the tool penetrating feedrate.

If not programmed or programmed with a value of "0", the CNC will assume 50% of the feedrate in the plane (F).

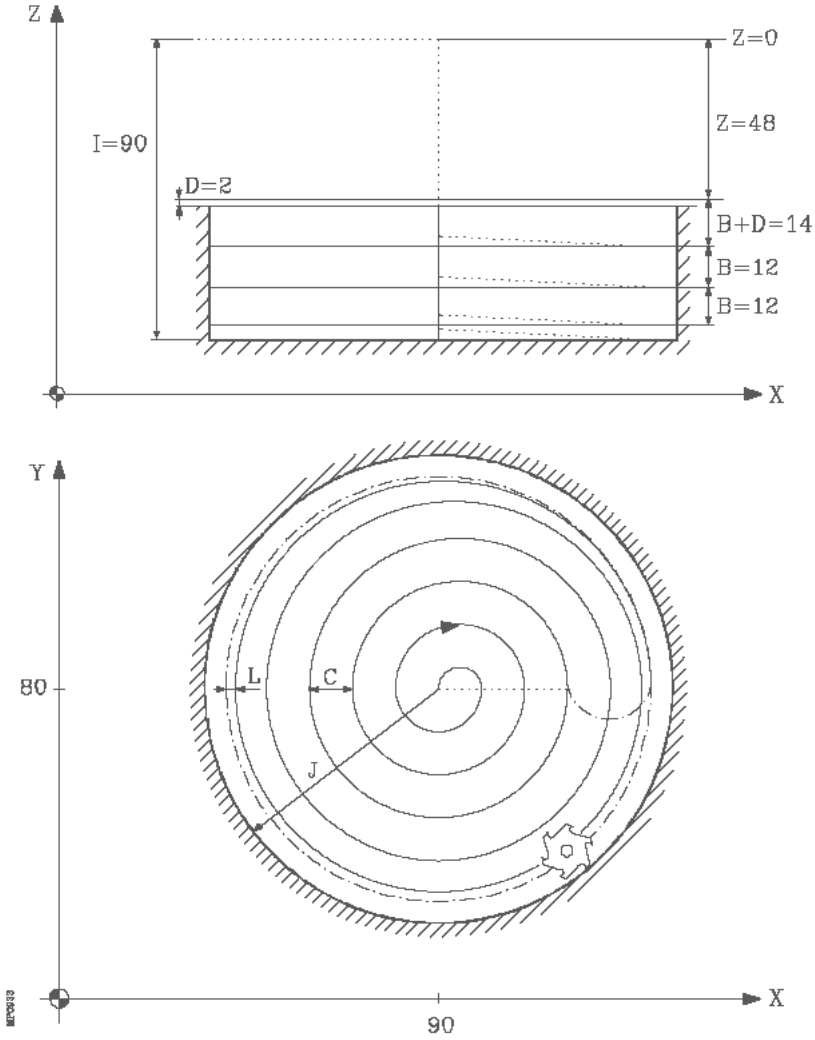




MP0503



Programming example assuming that the work plane is formed by the X and Y axes, that the longitudinal axis is the Z axis and that the starting point is X0 Y0 Z0:



(TOR1=6, TOT1=0)
T1 D1
M6
G0 G90 X0 Y0 Z0 ..... ; Starting point
G88 G98 G00 G90 X90 Y80 Z-48 I-90 J70 B12 C10
D2 H100 L5 V100 F300 S1000 T1 D1 M03 ..... Canned cycle definition
G80 ..... ; Canned cycle cancellation
G90 X0 Y0 ..... ; Positioning
M30 ..... ; End of program

9.5.10. G89. BORING CYCLE WITH WITHDRAWAL AT WORKING FEEDRATE (G01)

This cycle bores at the point indicated until the final programmed coordinate is reached.

It is possible to program a dwell at the bottom of the machined hole.

Working in cartesian coordinates, the basic structure of the block is as follows:

G89 G98/G99 X Y Z I K

G98 The tool withdraws to the Initial Plane, once the hole has been bored.

G99 The tool withdraws to the Reference Plane, once the hole has been bored.

XY±5.5 These are optional and define the movement of the axes of the main plane to position the tool at the machining point.

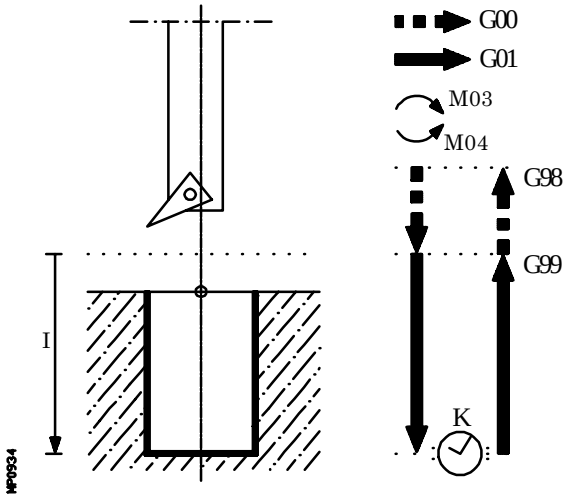
This point can be programmed in cartesian coordinates or in polar coordinates, and the coordinates may be absolute or incremental, along whether the machine is operating in G90 or G91.

Z±5.5 Defines the reference plane coordinate. It can be programmed in absolute coordinates or incremental coordinates and in this case will be referred to the initial plane.

If this is not programmed, the CNC will take the position occupied by the tool at that moment as the reference plane.

I±5.5 Defines boring depth. It can be programmed in absolute coordinates or incremental coordinates and in this case will be referred to the reference plane.

K5 Defines the dwell time, in hundredths of a second, after each boring step, until the withdrawal begins. Should this not be programmed, the CNC will take a value of K0.



Basic operation

1. If the spindle was in operation previously, its turning direction is maintained. If it was not in movement, it will start by turning clockwise (M03).

2. Rapid movement of the longitudinal axis from the initial plane to the reference plane.

3. Movement at the working feedrate (G01) of the longitudinal axis to the bottom of the machined hole, and boring.

4. Spindle stop (M05).

5. Withdrawal at working feedrate of the longitudinal axis to the reference plane.

6. Withdrawal, at rapid feedrate (G00), of the longitudinal axis as far as the initial plane if G98 has been programmed.

Programming example assuming that the work plane is formed by the X and Y axes, that the longitudinal axis is the Z axis and that the starting point is X0 Y0 Z0:

```
T1
M6
G0 G90 X0 Y0 Z0 ..... ;Starting point
G89 G98 G91 X250 Y350 Z-98 I-22 K20 F100 S500 .. ;Canned cycle definition
G80 ..... ;Canned cycle cancellation
G90 X0 Y0 ..... ;Positioning
M30 ..... ;End of program
```

10. *MULTIPLE MACHINING*

Multiple functions are defined as a series of functions which allow a machining operation to be repeated along a given path.

The programmer will select the type of machining, which can be a canned cycle or a subroutine (which must be programmed as a modal subroutine) defined by the user.

Machining subroutines are defined by the following functions:

G60: multiple machining in a straight line pattern.
G61: multiple machining in a rectangular pattern.
G62: multiple machining in a grid pattern.
G63: multiple machining in a circular pattern.
G64: multiple machining in an arc pattern.
G65: multiple machining in an arc-chord pattern.

These functions can be performed on any work plane and must be defined every time they are used, as they are not modal.

It is absolutely essential for the machining which it is required to repeat to be active. In other words, these functions will only make sense if they are under the influence of a canned cycle or under the influence of a modal subroutine.

To perform multiple machining, follow these steps:

1.- Move the tool to the first point of the multiple machining operation.

2- Define the canned cycle or modal subroutine to be repeated at all the points.

3.- Define the multiple operation to be performed.

All machining operations programmed with these functions will be done under the same working conditions (T,D,F,S) which were selected when defining the canned cycle or modal subroutine.

Once the multiple machining operation has been performed, the program will recover the history it had before starting this machining, even when the canned cycle or modal subroutine will remain active. Now feedrate F corresponds to the feedrate programmed for the canned cycle or modal subroutine.

Likewise, the tool will be positioned at the last point where the programmed machining operation was done.

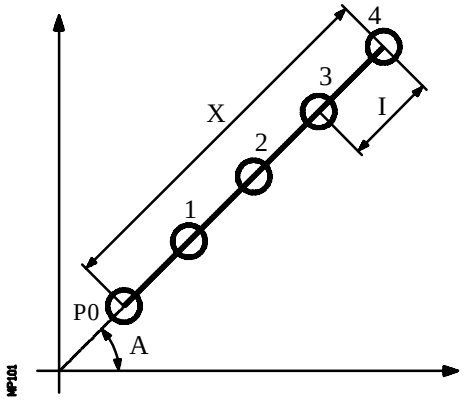
If multiple machining of a modal subroutine is performed in the Single Block mode, this subroutine will be performed complete (not block by block) after each programmed movement.

A detailed explanation is given on the next page of multiple machining operations, assuming in each case, that the work plane is formed by X and Y axes.

10.1 G60: MULTIPLE MACHINING IN A STRAIGHT LINE PATTERN

The programming format of this cycle is as follows:

G60	**A**	**X**	**I**	**P**	**Q**	**R**	**S**	**T**	**U**	**V**
		X	**K**							
		I	**K**							



A(+/-5.5) Defines the angle which forms the machining path with the abscissa axis. It is expressed in degrees and if not programmed, the value A=0 will be taken.

X(5.5) Defines the length of the machining path.

I(5.5) Defines the pitch between machining operations.

K(5) Defines the number of total machining operations in the section, including the machining definition point.

Due to the fact that machining may be defined with any two points of the **X I K** group, the CNC allows the following definition combinations: **XI**, **XK**, **IK**.

Nevertheless, if format **XI** is defined, care should be taken to ensure that the number of machining operations is an integer number, otherwise the CNC will show the corresponding error code.

P,Q,R,S,T,U,V These parameters are optional and are used to indicate at which points or between which those programmed points it is not required to machine.

Thus, programming P7 indicates that it is not required to do machining at point 7, and programming Q10.013 indicates that machining is not required from point 10 to 13, or expressed in another way, that no machining is required at points 10, 11, 12 and 13.

When it is required to define a group of points (Q10.013), care should be taken to define the final point with three digits, as if Q10.13 is programmed, multiple machining understands Q10.130.

The programming order for these parameters is P Q R S T U V, it also being necessary to maintain the order in which the points assigned to these are numbered, i.e., the numbering order of the points assigned to Q must be greater than that assigned to P and less than that assigned to R.

Example:

Proper programming P5.006 Q12.015 R20.022
Improper programming P5.006 Q20.022 R12.015

If these parameters are not programmed, the CNC understands that it must perform machining at all the points along the programmed path.

Basic operation:

1. Multiple machining calculates the next point of those programmed where it is wished to machine.

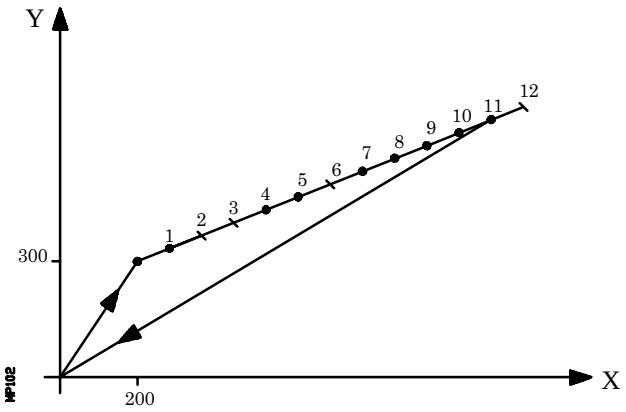
2. Rapid traverse (G00) to this point.

3. Multiple machining will perform the canned cycle or modal subroutine selected after this movement.

4. The CNC will repeat steps 1-2-3 until the programmed path has been completed.

   After completing multiple machining, the tool will be positioned at the last point along the programmed path where machining was performed.

Programming example assuming that the work plane is formed by the X and Y axes, that the longitudinal axis is Z and that the starting point is X0 Y0 Z0.




```
G81 G98 G00 G91 X200 Y300 Z-8 I-22 F100 S500   ;Canned cycle positioning and definition
G60 A30 X1200 I100 P2.003 Q6 R12               ;Defines multiple machining
G80                                            ;Cancels canned cycle
G90 X0 Y0                                      ;Positioning
M30                                            ;End of program
```

It is also possible to write the multiple machining definition block in the following ways:

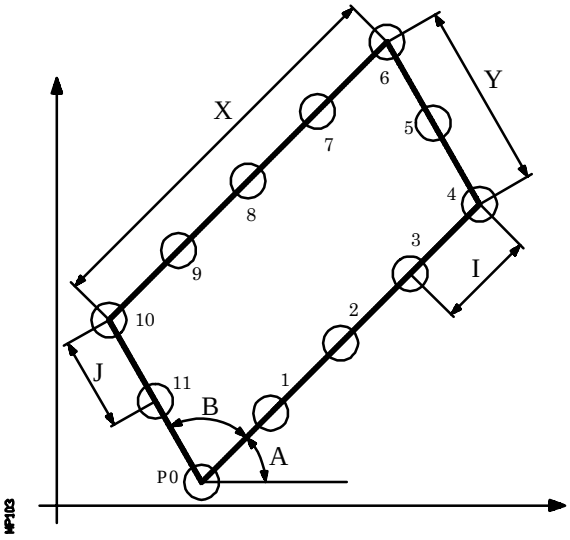
G60 A30 **X1200 K13** P2.003 Q6 R12

G60 A30 **I100 K13** P2.003 Q6 R12

10.2 G61: MULTIPLE MACHINING IN A RECTANGULAR PATTERN

The programming format of this cycle is as follows:

G61 A B | X I | Y J | P Q R S T U V
| X K | Y D |
| I K | J D |



A(+/-5.5) Defines the angle formed by the machining path with the abscissa axis. It is expressed in degrees and if not programmed, the value A=0 will be taken.

B(+/-5.5) Defines the angle formed by the two machining paths. It is expressed in degrees and if not programmed, the value B=90 will be taken.

X(5.5) Defines the length of the machining path according to the abscissa axis.

I(5.5) Defines the pitch between machining operations according to the abscissa axis.

K(5) Defines the number of total machining operations in the abscissa axis, including the machining definition point.

Due to the fact that machining may be defined according to the abscissa axis with any two points of the **X I K** group, the CNC allows the following definition combinations: **XI**, **XK**, **IK**.

Nevertheless, if format **XI** is defined, care should be taken to ensure that the number of machining operations is an integer number, otherwise the CNC will show the corresponding error code.

Y(5.5) Defines the length of the machining path according to the ordinate axis.

J(5.5) Defines the pitch between machining operations according to the ordinate axis.

D(5) Defines the number of total machining operations in the ordinate axis, including the machining definition point.

Due to the fact that machining may be defined according to the ordinate axis with any two points of the **Y J D** group, the CNC allows the following definition combinations: **YJ**, **YD**, **JD**.

Nevertheless, if format **YJ** is defined, care should be taken to ensure that the number of machining operations is an integer number, otherwise the CNC will show the corresponding error code.

P,Q,R,S,T,U,V These parameters are optional and are used to indicate at which points or between which of those programmed points it is not required to machine.

Thus, programming P7 indicates that it is not required to do machining at point 7, and programming Q10.013 indicates that machining is not required from point 10 to 13, or expressed in another way, that no machining is required at points 10, 11, 12 and 13.

When it is required to define a group of points (Q10.013), care should be taken to define the final point with three digits, as if Q10.13 is programmed, multiple machining understands Q10.130.

The programming order for these parameters is P Q R S T U V, it also being necessary to maintain the order in which the points assigned to these are numbered, i.e., the numbering order of the points assigned to Q must be greater than that assigned to P and less than that assigned to R.

Example:

Proper programming P5.006 Q12.015 R20.022
Improper programming P5.006 Q20.022 R12.015

If these parameters are not programmed, the CNC understands that it must perform machining at all the points along the programmed path.

Basic operation:

1. Multiple machining calculates the next point of those programmed where it is wished to machine.

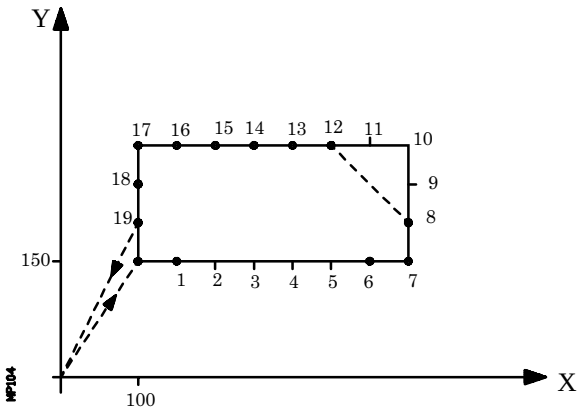
2. Rapid traverse (G00) to this point.

3. Multiple machining will perform the canned cycle or modal subroutine selected after this movement.

4. The CNC will repeat steps 1-2-3 until the programmed path has been completed.

   After completing multiple machining, the tool will be positioned at the last point along the programmed path where machining was performed.

Programming example assuming that the work plane is formed by the X and Y axes, that the longitudinal axis is Z and that the starting point is X0 Y0 Z0.

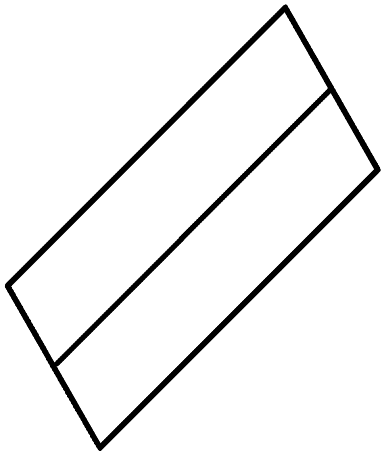



```
G81 G98 G00 G91 X100 Y150 Z-8 I-22 F100 S500   ;Canned cycle positioning and definition
G61 X700 I100 Y180 J60 P2.005 Q9.011           ;Defines multiple machining
G80                                            ;Cancels canned cycle
G90 X0 Y0                                      ;Positioning
M30                                            ;End of program
```

It is also possible to write the multiple machining definition block in the following ways:

G61 **X700 K8 J60 D4** P2.005 Q9.001

G61 **I100 K8 Y180 D4** P2.005 Q9.011



D(5) Defines the number of total machining operations in the ordinate axis, including the machining definition point.

Due to the fact that machining may be defined according to the ordinate axis with any two points of the **Y J D** group, the CNC allows the following definition combinations: **YJ**, **YD**, **JD**.

Nevertheless, if format **YJ** is defined, care should be taken to ensure that the number of machining operations is an integer number, otherwise the CNC will show the corresponding error code.

P,Q,R,S,T,U,V These parameters are optional and are used to indicate at which points or between which of those programmed points it is not required to machine.

Thus, programming P7 indicates that it is not required to do machining at point 7, and programming Q10.013 indicates that machining is not required from point 10 to 13, or expressed in another way, that no machining is required at points 10, 11, 12 and 13.

When it is required to define a group of points (Q10.013), care should be taken to define the final point with three digits, as if Q10.13 is programmed, multiple machining understands Q10.130.

The programming order for these parameters is P Q R S T U V, it also being necessary to maintain the order in which the points assigned to these are numbered, i.e., the numbering order of the points assigned to Q must be greater than that assigned to P and less than that assigned to R.

Example:

Proper programming P5.006 Q12.015 R20.022
Improper programming P5.006 Q20.022 R12.015

If these parameters are not programmed, the CNC understands that it must perform machining at all the points along the programmed path.

Basic operation:

1. Multiple machining calculates the next point of those programmed where it is wished to machine.

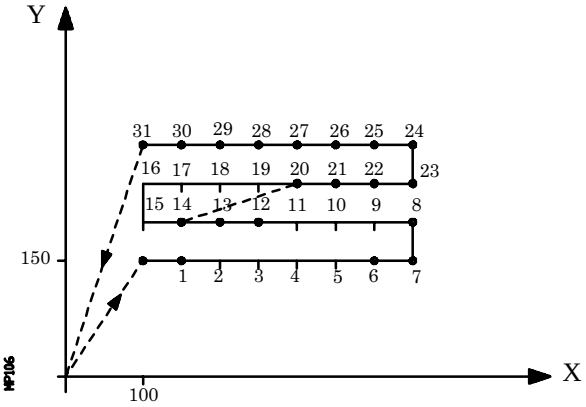
2. Rapid rapid traverse (G00) to this point.

3. Multiple machining will perform the canned cycle or modal subroutine selected after this movement.

4. The CNC will repeat steps 1-2-3 until the programmed path has been completed.

   After completing multiple machining, the tool will be positioned at the last point along the programmed path where machining was performed.

Programming example assuming that the work plane is formed by the X and Y axes, that the longitudinal axis is Z and that the starting point is X0 Y0 Z0.




```
G81 G98 G00 G91 X100 Y150 Z-8 I-22 F100 S500   ;Canned cycle positioning and definition
G62 X700 I100 Y180 J60 P2.005 Q9.011 R15.019   ;Defines multiple machining
G80                                            ;Cancels canned cycle
G90 X0 Y0                                      ;Positioning
M30                                            ;End of program
```

It is also possible to write the multiple machining definition block in the following ways:

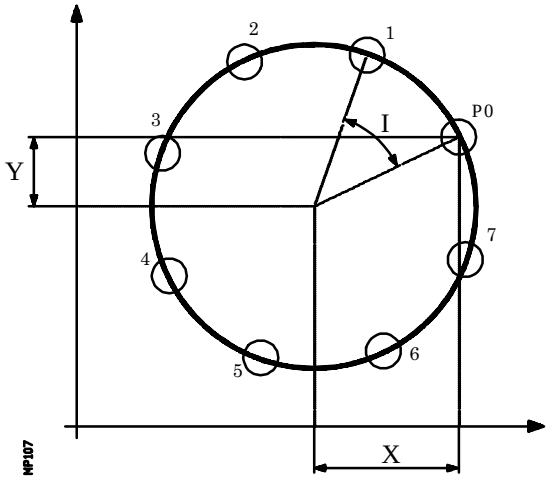
G61 **X700 K8 J60 D4** P2.005 Q9.001 R15.019

G61 **I100 K8 Y180 D4** P2.005 Q9.011 R15.019

10.4 G63: MULTIPLE MACHINING IN A CIRCULAR (BOLT-HOLE) PATTERN

The programming format of this cycle is as follows:

G63 X Y | I / K | C F P Q R S T U V



X(+/-5.5) Defines the distance from the starting point to the center along the abscissa axis.

Y(+/-5.5) Defines the distance from the starting point to the center along the ordinate axis.

With parameters X and Y the center of the circle is defined in the same way that I and J do this in circular interpolations (G02, G03).

I(+/-5.5) Defines the pitch angle between machining operations, if G00 or G01, the sign indicates the direction, "+" counter-clockwise, "-" clockwise.

K(5) Defines the number of total machining operations along the circle, including the machining definition point.

It will be enough to program I or K in the multiple machining definition block. Nevertheless, if K is programmed in a multiple machining operation in which movement between points is made in G00 or G01, machining will be done in the counter-clockwise direction.

C Indicates how movement is made between machining points. If it is not programmed, the value C=0 will be taken.

C=0: Movement is made in rapid feedrate (G00)
C=1: Movement is made in linear interpolation (G01).
C=2: Movement is made in clockwise circular interpolation (G02)
C=3: Movement is made in counter-clockwise circular interpolation (G03)

F(5.5) Defines the feedrate which is used for moving between points. Obviously, it will only apply for "C" values other than zero. If it is not programmed, the value F0 will be taken, maximum feedrate selected by the "MAXFEED" axis machine parameter.

P,Q,R,S,T,U,V These parameters are optional and are used to indicate at which points or between which of those programmed points it is not required to machine.

Thus, programming P7 indicates that it is not required to do machining at point 7, and programming Q10.013 indicates that machining is not required from point 10 to 13, or expressed in another way, that no machining is required at points 10, 11, 12 and 13.

When it is required to define a group of points (Q10.013), care should be taken to define the final point with three digits, as if Q10.13 is programmed, multiple machining understands Q10.130.

The programming order for these parameters is P Q R S T U V, it also being necessary to maintain the order in which the points assigned to these are numbered, i.e., the numbering order of the points assigned to Q must be greater than that assigned to P and less than that assigned to R.

Example:

Proper programming P5.006 Q12.015 R20.022
Improper programming P5.006 Q20.022 R12.015

If these parameters are not programmed, the CNC understands that it must perform machining at all the points along the programmed path.

Basic operation:

1. Multiple machining calculates the next point of those programmed where it is wished to machine.

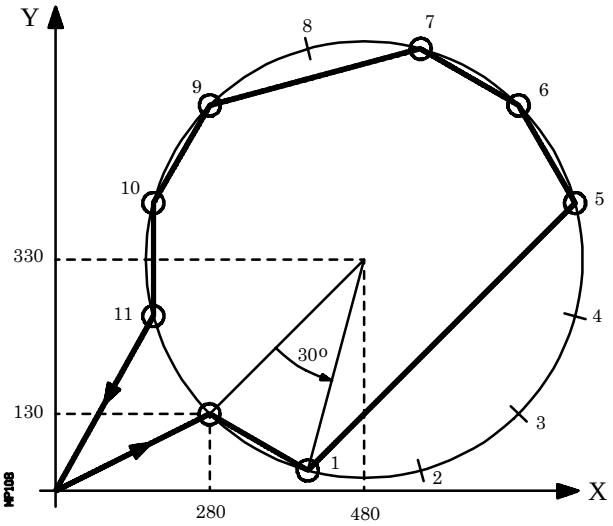
2. Movement programmed by "C" (G00,G01,G02 or G03) to this point.

3. Multiple machining will perform the canned cycle or modal subroutine selected after this movement.

4. The CNC will repeat steps 1-2-3 until the programmed path has been completed.

   After completing multiple machining, the tool will be positioned at the last point along the programmed path where machining was performed.

   Programming example assuming that the work plane is formed by the X and Y axes, that the longitudinal axis is Z and that the starting point is X0 Y0 Z0.




```
G81 G98 G01 G91 X280 Y130 Z-8 I-22 F100 S500   ;Canned cycle positioning and definition
G63 X200 Y200 I30 C1 F200 P2.004 Q8            ;Defines multiple machining
G80                                            ;Cancels canned cycle
G90 X0 Y0                                      ;Positioning
M30                                            ;End of program
```

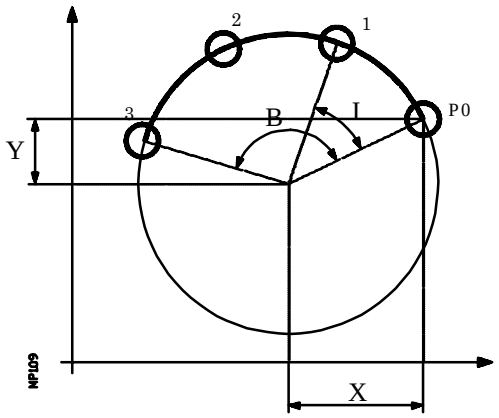
It is also possible to write the multiple machining definition block in the following ways:

G63 X200 Y200 **K12** C1 F200 P2.004 Q8

10.5 G64: MULTIPLE MACHINING IN AN ARC PATTERN

The programming format of this cycle is as follows:

G64 X Y B | I / K | C F P Q R S T U V



X(+/-5.5) Defines the distance from the starting point to the center along the abscissa axis.

Y(+/-5.5) Defines the distance from the starting point to the center along the ordinate axis.

With parameters X and Y the center of the circle is defined in the same way that I and J do this in circular interpolations (G02, G03).

B(5.5) Defines the angular stroke of the machining path and is expressed in degrees.

I(+/-5.5) Defines the pitch angle between machining operations, if G00 or G01, the sign indicates the direction, "+" counter-clockwise, "-" clockwise.

K(5) Defines the number of total machining operations along the circle, including the machining definition point.

It will be enough to program I or K in the multiple machining definition block. Nevertheless, if K is programmed in a multiple machining operation in which movement between points is made in G00 or G01, machining will be done in the counter-clockwise direction.

C Indicates how movement is made between machining points. If it is not programmed, the value C=0 will be taken.

C=0: Movement is made in rapid feedrate (G00)
C=1: Movement is made in linear interpolation (G01).
C=2: Movement is made in clockwise circular interpolation (G02)
C=3: Movement is made in counter-clockwise circular interpolation (G03)

F(5.5) Defines the feedrate which is used for moving between points. Obviously, it will only have value for "C" values other than zero. If it is not programmed, the value F0 will be taken, maximum feedrate selected by the "MAXFEED" axis machine parameter.

P,Q,R,S,T,U,V These parameters are optional and are used to indicate at which points or between which of those programmed points it is not required to machine.

Thus, programming P7 indicates that it is not required to do machining at point 7, and programming Q10.013 indicates that machining is not required from point 10 to 13, or expressed in another way, that no machining is required at points 10, 11, 12 and 13.

When it is required to define a group of points (Q10.013), care should be taken to define the final point with three digits, as if Q10.13 is programmed, multiple machining understands Q10.130.

The programming order for these parameters is P Q R S T U V, it also being necessary to maintain the order in which the points assigned to these are numbered, i.e., the numbering order of the points assigned to Q must be greater than that assigned to P and less than that assigned to R.

Example:

Proper programming P5.006 Q12.015 R20.022
Improper programming P5.006 Q20.022 R12.015

If these parameters are not programmed, the CNC understands that it must perform machining at all the points along the programmed path.

Basic operation:

1. Multiple machining calculates the next point of those programmed where it is wished to machine.

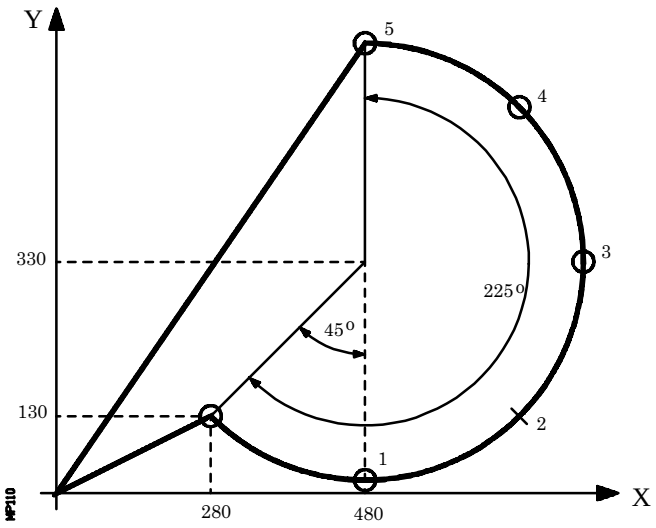
2. Movement programmed by "C" (G00,G01,G02 or G03) to this point.

3. Multiple machining will perform the canned cycle or modal subroutine selected after this movement.

4. The CNC will repeat steps 1-2-3 until the programmed path has been completed.

   After completing multiple machining, the tool will be positioned at the last point along the programmed path where machining was performed.

   Programming example assuming that the work plane is formed by the X and Y axes, that the longitudinal axis is Z and that the starting point is X0 Y0 Z0.



G81 G98 G01 G91 X280 Y130 Z-8 I-22 F100 S500	;Canned cycle positioning and definition
G64 X200 Y200 B225 K6 C3 F200 P2	;Defines multiple machining
G80	;Cancels canned cycle
G90 X0 Y0	;Positioning
M30	;End of program

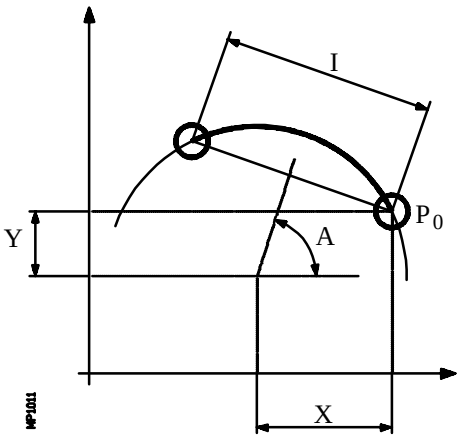
It is also possible to write the multiple machining definition block in the following ways:

G64 X200 Y200 B225 **K6** C3 F200 P2

10.6 G65: MACHINING PROGRAMMED BY MEANS OF AN ARC CHORD

This function allows activated machining to be performed at a point programmed by means of an arc chord. Only one machining operation will be performed, its programming format being:

G65 X Y | A / I | C F



X(+/-5.5) Defines the distance from the starting point to the center along the abscissa axis.

Y(+/-5.5) Defines the distance from the starting point to the center along the ordinate axis.

With parameters X and Y the center of the circle is defined in the same way that I and J do this in circular interpolations (G02, G03).

A(+/-5.5) Defines the angle formed by the perpendicular bisector of the chord with the abscissa axis and is expressed in degrees.

I(+/-5.5) Defines the chord length. When moving in G00 or G01, the sign indicates the direction, "+" counter-clockwise, "-" clockwise.

C Indicates how movement is made between machining points. If it is not programmed, the value C=0 will be taken.

C=0: Movement is made in rapid feedrate (G00)
C=1: Movement is made in linear interpolation (G01).
C=2: Movement is made in clockwise circular interpolation (G02)
C=3: Movement is made in counter-clockwise circular interpolation (G03)

F(5.5) Defines the feedrate which is used for moving between points. Obviously, it will only apply for "C" values other than zero. If it is not programmed, the value F0 will be taken, maximum feedrate selected by the "MAXFEED" axis machine parameter.

Basic operation:

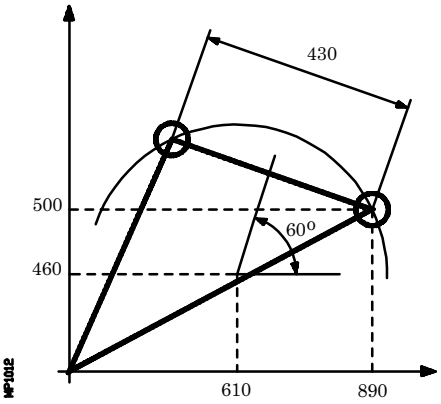
1. Multiple machining calculates the next point of those programmed where it is wished to machine.

2. Movement programmed by "C" (G00,G01,G02 or G03) to this point.

3. Multiple machining will perform the canned cycle or modal subroutine selected after this movement.

   After completing multiple machining, the tool will be positioned at the programmed point.

   Programming example assuming that the work plane is formed by the X and Y axes, that the longitudinal axis is Z and that the starting point is X0 Y0 Z0.



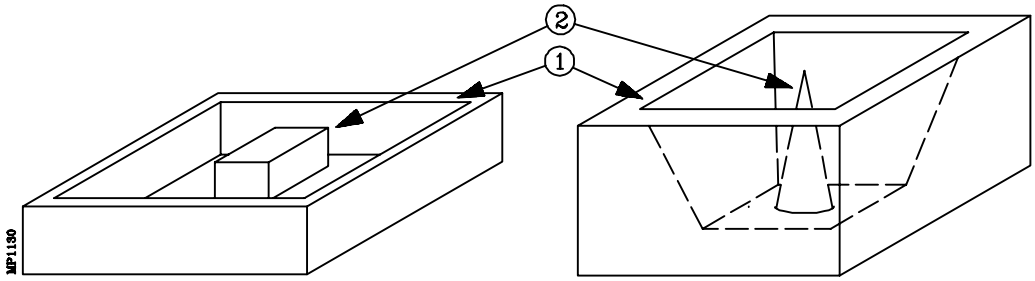
G81 G98 G01 G91 X890 Y500 Z-8 I-22 F100 S500	;Canned cycle positioning and definition
G65 X280 Y-40 A60 C1 F200	;Defines multiple machining
G80	;Cancels canned cycle
G90 X0 Y0	;Positioning
M30	;End of program

It is also possible to write the multiple machining definition block in the following ways:

G65 X-280 Y40 **I430** C1 F200

11. *IRREGULAR POCKET CANNED CYCLE (WITH ISLANDS)*

A pocket is composed by an external contour or profile (1) and a series of internal contours or profiles (2). These internal profiles are called islands.

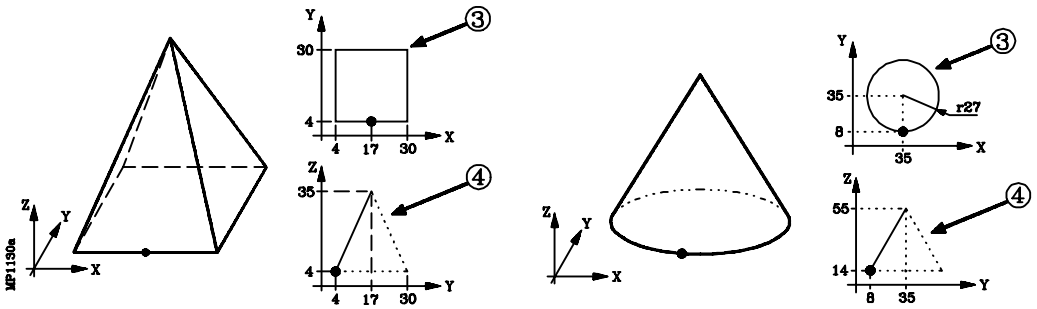


With this pocket canned cycle, 2D and 3D pockets may be machined.

2D pocket (Upper left-hand illustration).
Its inside and outside walls are vertical.
Its programming is detailed in the first part of this chapter.
To define the contours of a 2D pocket, the plane profile for all the contours must be defined.

3D pocket (Upper right-hand illustration).
When any of the inside or outside profiles and/or islands is not vertical.
Its programming is detailed in the second part of this chapter.

To define the contours of a 2D pocket, the plane profile (3) and the depth profile (4) for all the contours must be defined (even if they are vertical).



The call function for a 2D or 3D irregular pocket canned cycle is G66.

The machining of a pocket may consist of the following operations:

Drilling operation, prior to machining ..... Only on 2D pockets
Roughing operation ..... 2D and 3D pockets
Semi-finishing operation ..... Only on 3D pockets
Finishing operation ..... 2D and 3D pockets

11.1 2D POCKETS

The G66 function is not modal, therefore it must be programmed whenever it is required to perform a 2D pocket.

In a block defining an irregular pocket canned cycle, no other function can be programmed, its structure definition being:

G66 D H R I F K S E Q

D (0-9999) & H (0-9999) Label number of the first block (D) and last block (H) defining the drilling operation.
When not setting "H" only block "D" is executed.
When not setting "D" there is no drilling operation.

R (0-9999) & I (0-9999) Label number of the first block (R) and last block (I) defining the roughing operation.
When not setting "I" only block "R" is executed.
When not setting "R" there is no roughing operation.

F (0-9999) & K (0-9999) Label number of the first block (F) and last block (K) defining the finishing operation.
When not setting "K" only block "F" is executed.
When not setting "F" there is no finishing operation.

S (0-9999) & E (0-9999) Label number of the first block (S) and last block (E) defining the geometry of the profiles forming the pocket.
Both parameters must be set.

Q (0-9999) Number of the program containing the geometry definition, parameters S and E.
If it is in the same program, "Q" need not be defined.

Programming example:

```
      G00 G90 X100 Y200 Z50 F5000 T1 D2  ;Initial positioning
      M06
      G66 D100 R200 I210 F300  S400 E500  ;Definition of irregular pocket canned cycle
      M30                                 ;End of program

N100  G81 .....                           ;Defines the drilling operation
N200  .....                               ;Starts the roughing operation
      G67 .....
N210  .....                               ;End the roughing operation
N300  G68 .....                           ;Defines the finishing operation
N400  G0 G90 X300 Y50 Z3                  ;Starts the geometry description
      .....
      .....
N500  G2 G6 X300 Y50 I150 J0              ;End of geometry description
```

Basic operation:

1.- Drilling operation. Only if it has been programmed.

After analyzing the geometry of the pocket with islands, the tool radius and the angle of the path programmed in the roughing operation, the CNC will calculate the coordinates of the point where the selected drilling operation must be performed.

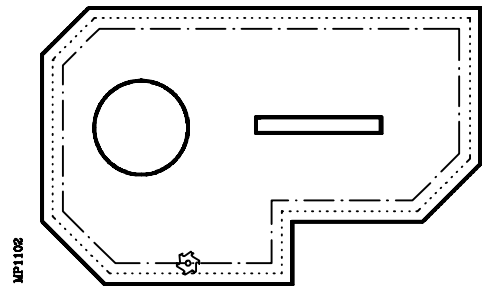
2.- Roughing operation. Only if it has been programmed.

It consists of several surface milling passes, until the total depth programmed has been reached. On each surface milling pass, the steps below will be followed depending on the type of machining that has been programmed:

Case A: When the machining paths are linear and maintain a certain angle with the abscissa axis.

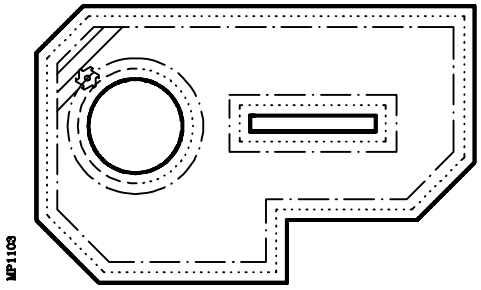
* It first contours the external profile of the part.

  If the finishing operation has been selected on the cycle call, this contouring is performed leaving the finishing stock programmed for the finishing pass.

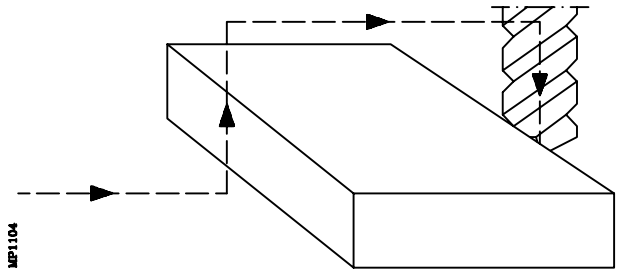


* Next the milling operation, with the programmed feed and steps.

  If, while milling, an island is run into for the first time, it will be contoured.

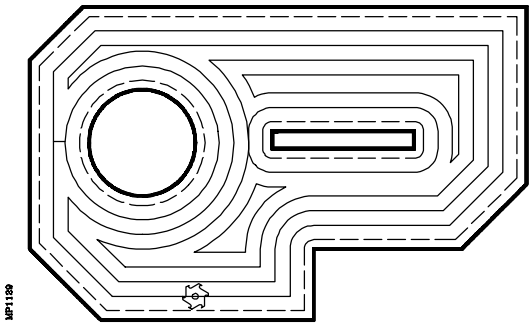


After the contouring and the remaining times, the tool will pass over the island, withdrawing along the longitudinal axis, to the reference plane, and will continue machining once the island has been cleared.



Case B: When the machining paths are concentric

* The roughing operation is carried out along paths concentric to the profile. The machining will be done as fast as possible avoiding (when possible) going over the islands.

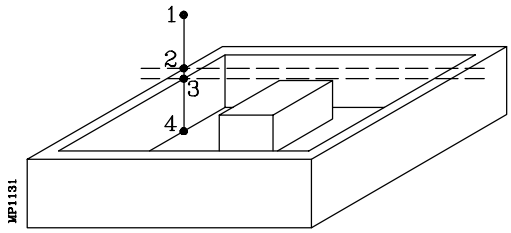


3.- Finishing operation. Only if it has been programmed.

This operation can be done on a single pass or on several, as well as following the profiles in the programmed direction or in the opposite.

The CNC will machine both the external profile and the islands, making tangential approaches and exits to these with a constant surface speed.

In the pocket canned cycle with islands, there are four coordinates along the longitudinal axis (selected with G15), which, due to their importance, are discussed below:



1.- Initial plane coordinate. This coordinate is given by the position which the tool occupies when the cycle is called.

2.- Reference plane coordinate. This represents an approach coordinate to the part, and must be programmed in absolute coordinates.

3.- Part surface coordinate. This is programmed in absolute coordinates and in the first profile definition block.

4.- Machining depth coordinate. This is programmed in absolute coordinates.

Conditions after finishing the cycle

Once the canned cycle has been completed, the active feedrate will be the last programmed feedrate, the one relating to the roughing or finishing operation. Likewise, the CNC will assume functions G00, G07, G40 and G90.

11.1.1 DRILLING OPERATION

This operation is optional and in order to be executed it is necessary to also program a roughing operation.

It is mainly used when the tool programmed in the roughing operation does not machine along the longitudinal axis, allowing, by means of this operation, the access of this tool to the surface to be roughed off.

It will be programmed in a block which will need to bear a label number in order to indicate to the canned cycle the block where the drilling operation is defined.

```
Example: G66 D100 R200 F300 S400 E500  ; Definition of the irregular pocket cycle.
         N100 G81 .....                ; Definition of the drilling operation.
```

The drilling canned cycles that can be programmed are:

- **G69** Complex deep hole drilling canned cycle (with variable step).
- **G81** Drilling canned cycle.
- **G82** Drilling canned cycle with dwell.
- **G83** Simple deep hole drilling canned cycle (with constant step).

When defining the drilling operation, the corresponding definition parameters must be programmed together with the required function.

In a block of this type, only cycle definition parameters must be programmed, without defining XY positioning, as the canned cycle itself will calculate the coordinate of the point or points to be drilled according to the programmed profile and the roughing angle.

After the definition parameters, auxiliary F S T D M functions can be programmed, if so wished. No M function can be programmed if it has an associated subroutine.

It is possible to program the M06 function in this block (if it does not have an associated subroutine), to make the tool change. Otherwise, the CNC will show the corresponding error. If the M06 has an associated subroutine, the drilling tool “T” must be selected before calling the cycle.

Examples:

```
N100 G69 G98 G91 Z-4 I-90 B1.5 C0.5 D2 H2 J4 K100 F500 S3000 M3
N120 G81 G99 G91 Z-5 I-30 F400 S2000 T3 D3 M3
N220 G82 G99 G91 Z-5 I-30 K100 F400 S2000 T2 D2 M6
N200 G83 G98 G91 Z-4 I-5 J6 T2 D4
```

11.1.2 ROUGHING OPERATION

This is the main operation in the machining of an irregular pocket, and its programming is optional.

This operation will be carried out in either square corner (G07) or round corner (G05) as it is currently selected. However, the canned cycle will assign the G07 format to the necessary movements.

It will be programmed in a block which will need to bear a label number in order to indicate to the canned cycle the block where the roughing operation is defined.

Example: G66 D100 **R200** F300 S400 E500 ; Definition of the irregular pocket cycle.
N200 G67 ..... ; Definition of the roughing operation.

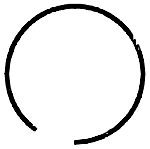
The function for the roughing operation is G67 and its programming format:

G67 A B C I R K V F S T D M

A(+/-5.5) Defines the angle which forms the roughing path with the abscissa axis.

A

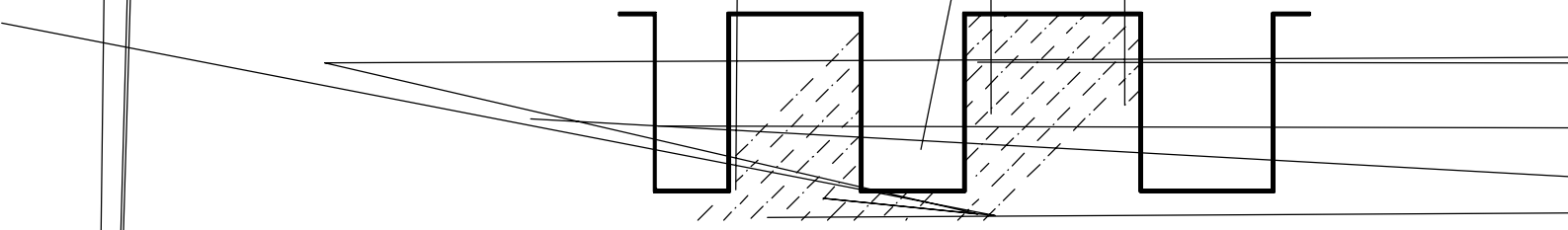
If parameter "A" is not programmed, the roughing operation is carried out following concentric paths. It will be machined as fast as possible since it does not have to go over the islands.



B(+/-5.5) Defines the machining pass along the longitudinal axis (depth of the roughing pass). It must be defined and it must have a value other than 0; otherwise, the roughing operation will be cancelled.

- If programmed with a positive sign, all the roughing will be performed with the same machining pass, and the canned cycle calculates a pass equal to or smaller than the programmed pass.

- If programmed with a negative sign, all the roughing will be performed with the programmed pass, and the canned cycle will adjust the last pass to obtain the total programmed depth.



C(+/-5.5) Defines the milling pass in roughing along the main plane, the entire pocket being performed with the given pass, and the canned cycle adjusts the last milling pass.

If it is not programmed or is programmed with either a value of 0, it will assume a value of 3/4 the diameter of the selected tool.

If programmed with a value greater than the tool diameter, the CNC will issue the corresponding error.

I(+/-5.5) Defines the total depth of the pocket and is programmed in absolute coordinates. It must be programmed.

R(+/-5.5) Defines the reference plane coordinate and is programmed in absolute coordinates. It must be programmed.**2D**

K(1) Defines the type of profile intersection to be used.

0 = Basic profile intersection.
1 = Advanced profile intersection.

If not programmed, a value of 0 will be assumed. Both intersection types will be discussed later on.

V (5.5) Defines the tool penetrating feedrate.

If not programmed or programmed with a value of "0", the CNC will assume 50% of the feedrate in the plane (F).

F (5.5) Optional. Defines the machining feedrate in the plane.

S (5.5) Optional. Defines the spindle speed.

T (4) Defines the tool used for the roughing operation. It must be programmed.

D (4) Optional. Defines the tool offset number.

M Optional. Up to 7 miscellaneous M functions can be programmed.

This operation allows M06 with an associated subroutine to be defined, and the tool change is performed before beginning the roughing operation.

11.1.3 FINISHING OPERATION

This is the last operation in the machining of an irregular pocket, and its programming is optional.

It will be programmed in a block which will need to bear a label number in order to indicate to the canned cycle the block where the finishing operation is defined.

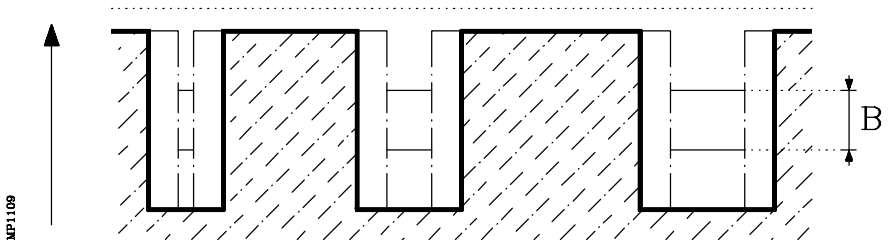
```
Example: G66 D100 R200 F300 S400 E500  ; Definition of the irregular pocket cycle.
         N300 G68 .....                ; Definition of the finishing operation.
```

The function for the finishing operation is G68 and its programming format:

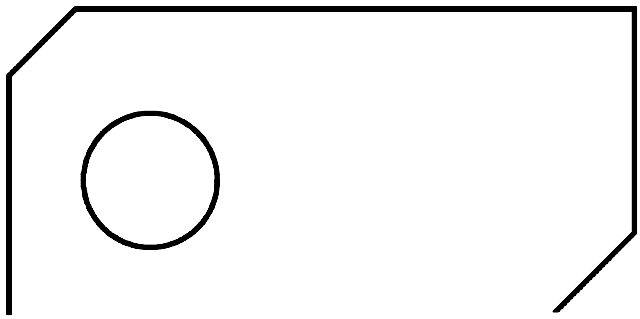
G68 B L Q I R K V F S T D M

B(±5.5) Defines the machining pass along the longitudinal axis (depth of the finishing pass).

- If it is programmed with a value of 0, the CNC will perform a single finishing pass with the total depth of the pocket.
- If programmed with a positive sign, all the roughing will be performed with the same machining pass, and the canned cycle calculates a pass equal to or lower than the programmed pass.
- If programmed with a negative sign, all the roughing will be performed with the programmed pass, and the canned cycle will adjust the last pass to obtain the total programmed depth.



L(±5.5) Defines the value of the finishing stock which it is required to leave on the side walls of the pocket before the finishing operation.



- If programmed with a positive value, the finishing pass will be carried out in square corner (G07).

- If programmed with a negative value, the finishing pass will be carried out in round corner (G05).

- If programmed with a value of 0, no finishing pass will be carried out.

Q Indicates the direction of the finishing pass.The finishing pass on the islands is always carried out in the opposite direction.

Q = 0 The finishing pass is carried out in the same direction as the outside profile was programmed.

Q = 1 The finishing pass is carried out in the opposite direction to the one programmed.

Q = 2 Reserved.

Any other value will generate the corresponding error message. If parameter "Q" is not programmed, the cycle assumes Q0.

I(±5.5) Defines the total depth of the island and it is given in absolute coordinates.

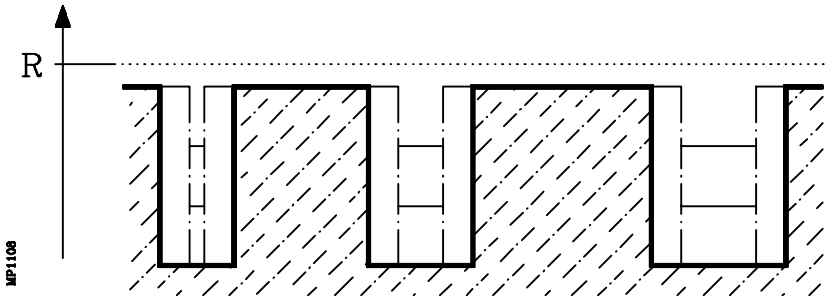
- If the island has a roughing operation, it is not necessary to define this parameter since it has been programmed in that operation. However, if programmed in both operations, the canned cycle will assume the particular depth indicated for each operation.

- If the island has no roughing operation, it is necessary to define this parameter.

R (±5.5) Defines the coordinate of the reference plane and it is given in absolute values.

- If the island has a roughing operation, it is not necessary to define this parameter since it has been programmed in that operation. However, if programmed in both operations, the canned cycle will assume the particular depth indicated for each operation.

- If the island has no roughing operation, it is necessary to define this parameter.



K(1) Defines the type of profile intersection to be used.

0 = Basic profile intersection.
1 = Advanced profile intersection.

If the island has a roughing operation, it is not necessary to define this parameter since it has been programmed in that operation. However, if programmed in both operations, the canned cycle will assume the one defined for the roughing operation.

If no roughing operation has been defined and this parameter is not programmed, the canned cycle will assume a K0 value. Both types of intersection are described later on.

V (5.5) Defines the tool penetrating feedrate.

If not programmed or programmed with a value of "0", the CNC will assume 50% of the feedrate in the plane (F).

F (5.5) Optional. Defines the machining feedrate in the plane.

S (5.5) Optional. Defines the spindle speed.

T (4) Defines the tool used for the roughing operation. It must be programmed.

D (4) Optional. Defines the tool offset number.

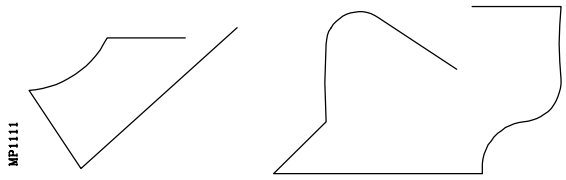
M Optional. Up to 7 miscellaneous M functions can be programmed.

This operation allows M06 with an associated subroutine to be defined, and the tool change is performed before beginning the roughing operation.

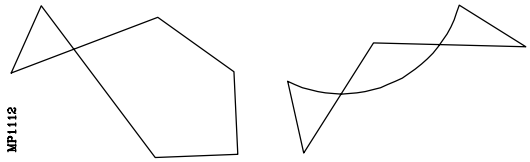
11.1.4 PROFILE PROGRAMMING RULES

When outside and inside profiles of an irregular pocket are programmed the following programming rules must be followed:

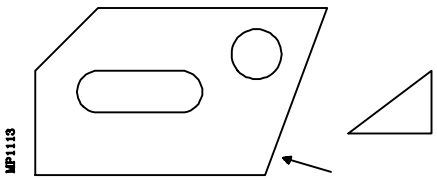
1.- All types of programmed profiles must be closed. The following examples cause a geometry error.



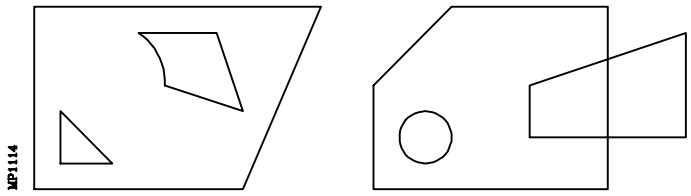
2.- No profile must intersect itself. The following examples cause a geometry error.



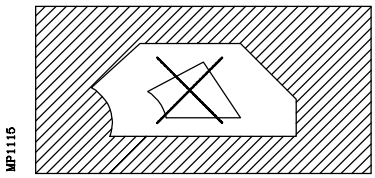
3.- When more than one outside profile has been programmed, the canned cycle assumes the one occupying the largest surface.



4.- It is not required to program inside profiles. Should these be programmed, they must be partially or totally internal with respect to the outside profile. Some examples are given below.



5.- An internal profile totally contained within another internal profile cannot be programmed. In this case, only the most external profile will be considered.



The canned cycle will verify all these geometry rules before beginning to make the pocket adapting the profile of the pocket to them and displaying the error message when necessary.

11.1.5 INTERSECTION OF PROFILES

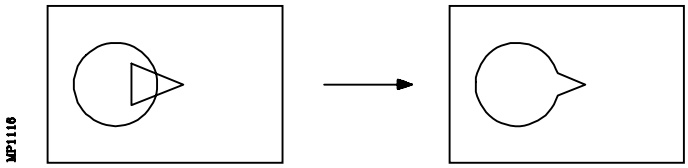
In order to facilitate the programming of profiles, the canned cycle allows the profiles to intersect one another and the external profile.

The two available types of intersection can be selected by parameter "**K**"

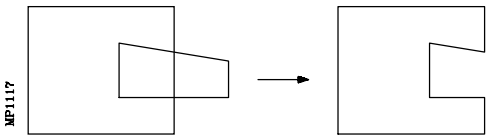
11.1.5.1 BASIC PROFILE INTERSECTION (K=0)

When selecting this type, the following profile intersecting rules are to be followed:

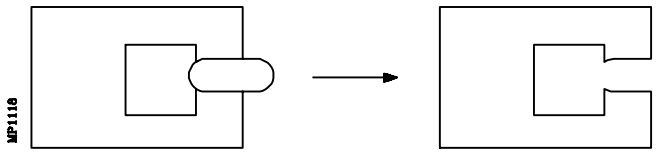
1.- The intersection of islands generates a new inside profile which is their boolean union. Example:



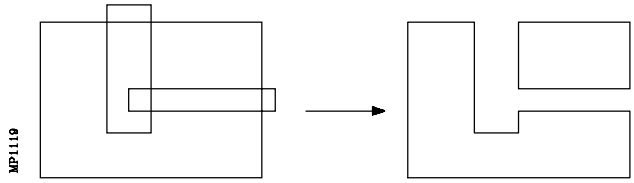
2.- The intersection between an internal and an external profile generates a new external profile as a result of the difference between the external and the internal profiles. Example:



3.- If there is an inside profile which has an intersection with another inside profile and with the external profile, the canned cycle first makes the intersection between the inside profiles and then the intersection of these with the external profile.



4.- As a result of the intersection of the inside profiles with the outside one, a single pocket will be obtained which corresponds to the outside profile having the largest surface. The rest will be ignored.



5.- If the finishing operation has been programmed, the profile of the resulting pocket must comply with all the tool compensation rules, since if a profile is programmed which cannot be machined by the programmed finishing tool, the CNC will show the corresponding error.

11.1.5.2 ADVANCED PROFILE INTERSECTION (K=1)

When selecting this type, the following profile intersecting rules are to be followed:

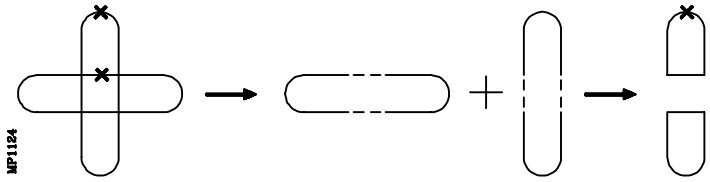
1.- The initial point of each contour determines the section to be selected.

In a profile intersection, each contour is divided into several lines that could be grouped as:

- Lines external to the other contour.
- Lines internal to the other contour.

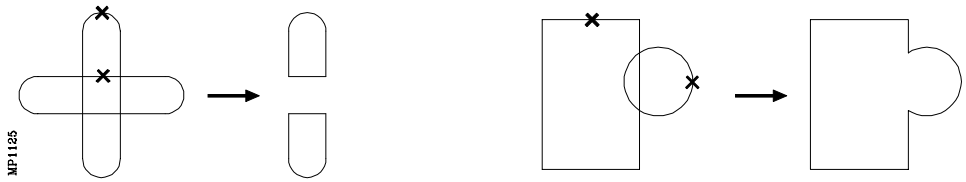
This type of profile intersection selects in each contour the group of lines where the profile defining point is included.

The following example shows the explained selection process. The solid lines indicate the lines external to the other contour and the dashes indicate the internal lines. The initial point of each contour is indicated with an "x".

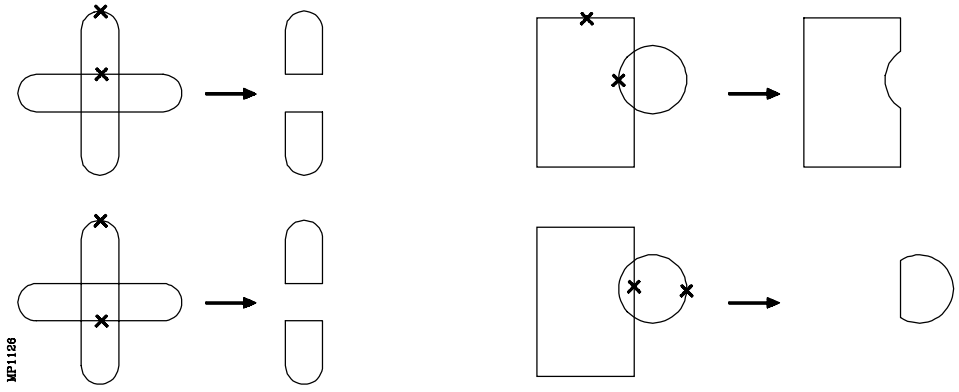


Examples of profile intersections:

Boolean Addition:



Boolean Subtraction:



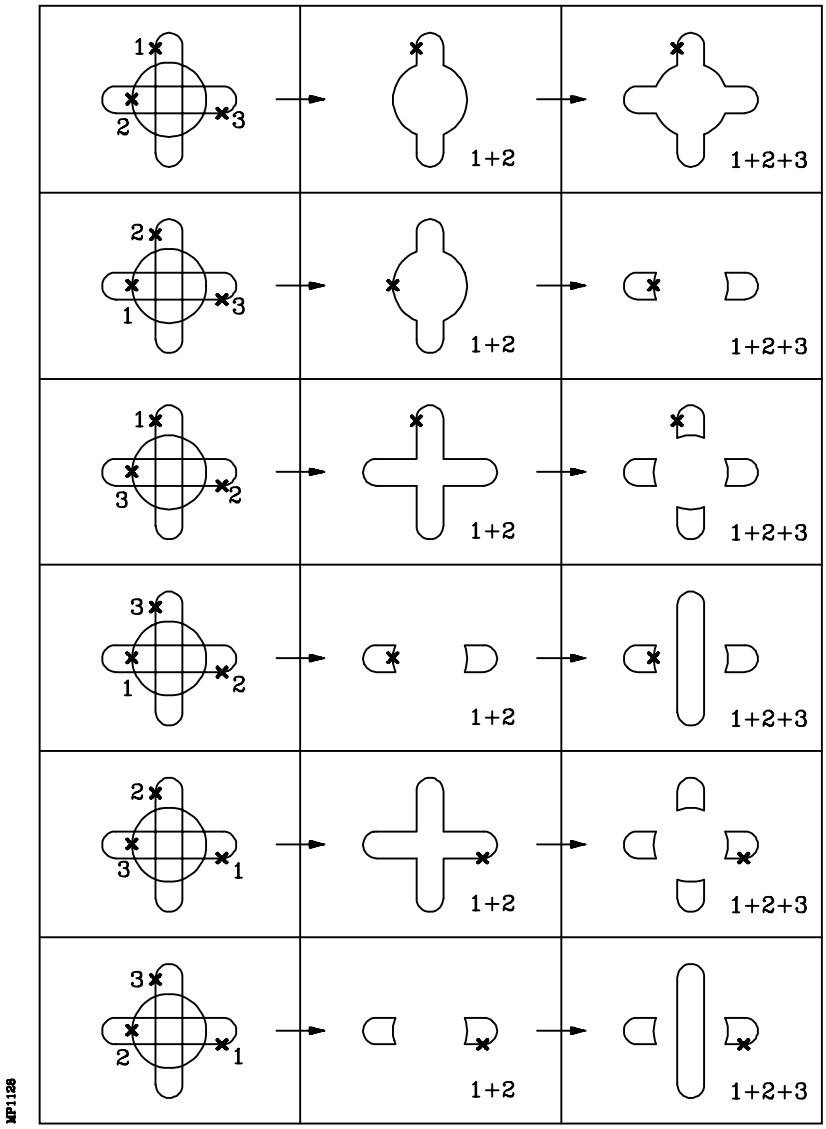
Boolean Intersection:

2.- The programming sequence for the different profiles is determinant when having an intersection of more than 2 profiles.

The profile intersection process is performed according to the order in which the profiles have been programmed. This way, the result of the intersection between the first two will be intersected with the third one and so forth.

The initial point of the resulting profiles always coincides with the initial point which defined the first profile.

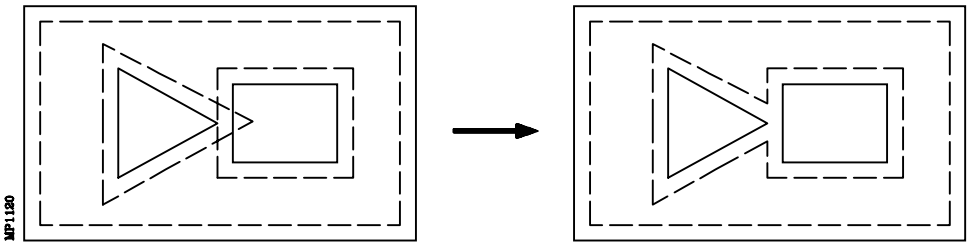
Examples:



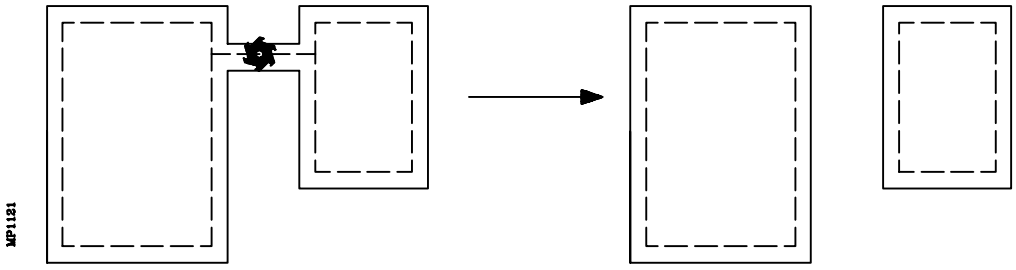
11.1.5.3 RESULTING PROFILE

Once the profiles of the pocket and islands have been obtained, the canned cycle calculates the remaining profiles according to the radius of the roughing tool and the programmed finishing stock.

It may occur that in this process intersections are obtained which do not appear among the programmed profiles. Example:



If there is an area in which the roughing tool cannot pass, when the intersection is made between the offset of the profiles, several pockets will be obtained as a result, all of which will be machined. Example:



11.1.6 PROFILE PROGRAMMING SYNTAX

The outside profile and the inside profiles or islands which are programmed must be defined by simple geometrical elements such as straight lines or arcs.

The first definition block (where the external profile starts) and the last (where the last profile defined ends) must be provided with the block label number. These label numbers will be those which indicate to the canned cycle the beginning and end of the geometric description of the profiles which make up the pocket.

Example:
```
G66 D100 R200 F300 S400 E500  ;Definition of irregular pocket
N400 G0 G90 X300 Y50 Z3       ;Beginning of geometric description
------ ----- ---- ---
N500 G2 G6 X300 Y50 I150 J0   ;End of geometric description
```

The profile programming syntax must comply with the following rules:

1.- The external profile must begin in the first definition block of the geometric description of the part profiles. This block will be assigned a label number in order to indicate canned cycle G66 the beginning of the geometric description.

2.- The part surface coordinate will be programmed in this block.

3.- All the internal profiles which are required may be programmed, one after the other. Each of these must commence with a block containing the **G00** function (indicating the beginning of the profile).

Warning:

Care must be taken to program G01, G02 or G03 in the block following the definition of the beginning, as G00 is modal, thus preventing the CNC from interpreting the following blocks as the beginnings of a new profile.

4.- Once the definition of the profiles has been completed, a label number must be assigned to the last block programmed, in order to indicate the canned cycle G66 the end of the geometric description.
Example

```
G0 G17 G90 X-350 Y0 Z50
G66 D100 R200 F300 S400 E500 ..... ;Description of cycle
G0 G90 X0 Y0 Z50
M30

N400 G0 G90 X-260 Y-190 Z4.5 ..... ;Beginning of first profile
     .....
     .....
     G0 X230 Y170 ..... ;Beginning of another profile
     G1.....
     .....
     G0 X-120 Y90 ..... ;Beginning of another profile
     G2.....
     .....
N500 G1 X-120 Y90 ..... ;End of geometric description
```

5.- Profiles are described as programmed paths, it being possible to include corner rounding, chamfers, etc., following the syntax rules defined for this purpose.

6.- Mirror images, scaling factor changes, rotation of coordinate system, zero offsets, etc., cannot be programmed in the description of profiles.

7.- Nor is it possible to program blocks in high level language, such as jumps, subroutine calls or parametric programming.

8.- Other canned cycles cannot be programmed.

In addition to the G00 function, which has a special meaning, the irregular pocket canned cycle allows the use of the following functions for the definition of profiles.

G01	Linear interpolation
G02	Clockwise circular interpolation
G03	Counter-clockwise circular interpolation
G06	Arc center in absolute coordinates
G08	Arc tangent to previous path.
G09	Arc defined by three points
G36	Controlled corner rounding
G39	Chamfer
G53	Programming with respect to machine reference zero (home)
G70	Programming in inches
G71	Programming in millimeters
G90	Absolute programming
G91	Incremental programming
G93	Polar origin preset

11.1.7 ERRORS

The CNC will issue the following errors:

ERROR 1023 : G67. Tool radius too large.
When selecting a wrong roughing tool.

ERROR 1024 : G68. Tool radius too large.
When selecting a wrong finishing tool.

ERROR 1025 : A tool of no radius has been programmed.
When using a tool with "0" radius while machining a pocket.

ERROR 1026 : A step greater than the tool diameter has been programmed.
When parameter "C" of the roughing operation is greater than the diameter of the roughing tool.

ERROR 1041 : A mandatory parameter not programmed in the canned cycle.
It comes up in the following instances:
- When parameters "I" and "R" have not been programmed in the roughing operation.
- When not using a roughing operation and not programming the "I" and "R" parameters for the finishing operation.

ERROR 1042 : Wrong canned cycle parameter value.
It comes up in the following instances:
- When parameter "Q" of the finishing operation has the wrong value.
- When parameter "B" of the finishing operation has a "0" value.
- When parameter "J" of the finishing operation has been programmed with a value greater than the finishing tool radius.

ERROR 1044 : The plane profile intersects itself in an irregular pocket with islands.
It comes up when any of the plane profiles of the programmed contours intersects itself.

ERROR 1046 : Wrong tool position prior to the canned cycle.
It comes up when calling the G66 cycle if the tool is positioned between the reference plane and the depth coordinate (bottom) of any of the operations.

ERROR 1047 : Open plane profile in an irregular pocket with islands.
It comes up when any of the programmed contours does not begin and end at the same point. It may be because G1 has not been programmed after the beginning, with G0, on any of the profiles.

ERROR 1048 : The part surface coordinate (top) has not been programmed in an irregular pocket with islands.
It comes up when the first point of the geometry does not include the pocket top coordinate.

ERROR 1049 : Wrong reference plane coordinate for the canned cycle.
It comes up when the coordinate of the reference plane is located between the part's "top" and "bottom" in any of the operations.

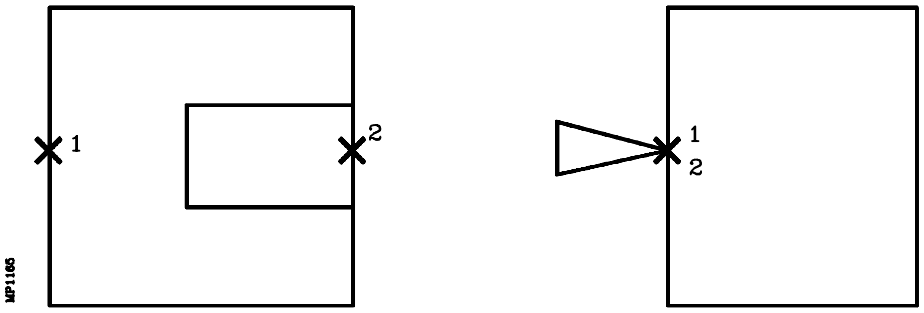
ERROR 1084 : Wrong circular path.

It comes up when any of the paths programmed in the geometry definition of the pocket is wrong.

ERROR 1227 : Wrong profile intersection in an irregular pocket with islands.

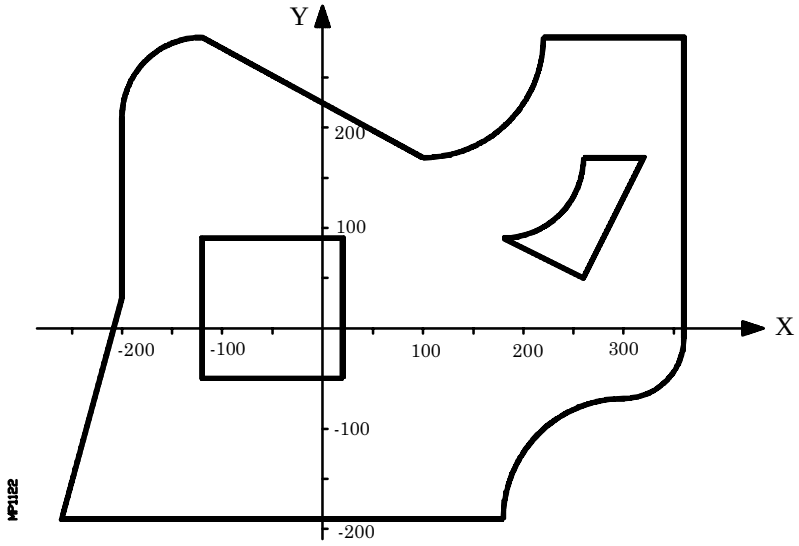
It comes up in the following instances:

- When two plane profiles have a common section (drawing on the left).
- When the initial points of two profiles in the main plane coincide (drawing on the right).



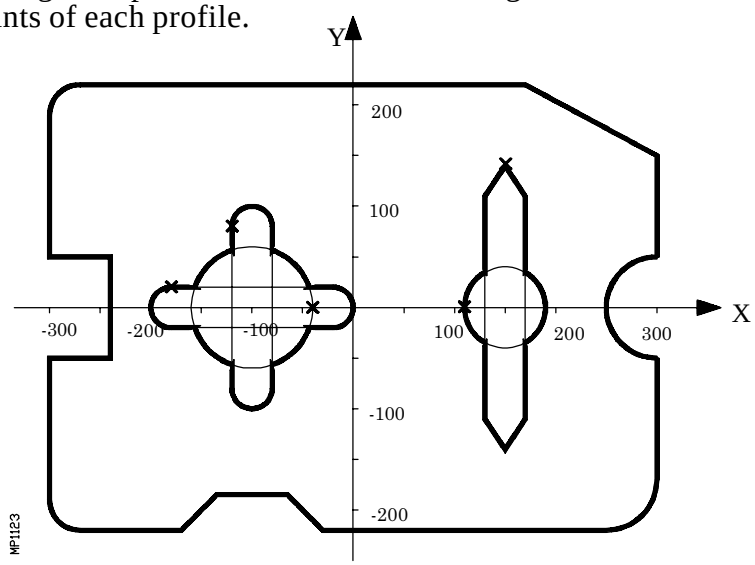
11.1.8 PROGRAMMING EXAMPLES

Programming example, without automatic tool changer




```
(TOR1=5, TOI1=0, TOL1=25,TOK1=0)    ;Tool 1 dimensions
(TOR2=3, TOI2=0, TOL2=20,TOK2=0)    ;Tool 2 dimensions
(TOR3=5, TOI3=0, TOL3=25,TOK3=0)    ;Tool 3 dimensions
G0 G17 G43 G90 X0 Y0 Z25 S800       ;Initial positioning
G66 D100 R200 F300 S400 E500        ;Irregular pocket description
M30                                 ;End of program

N100 G81 Z5 I-40 T3 D3 M6                          ;Definition of drilling operation
N200 G67 B20 C8 I-40 R5 K0 V100 F500 T1 D1 M6      ;Definition of roughing operation
N300 G68 B0 L0.5 Q0 V100 F300 T2 D2 M6             ;Definition of finishing operation

N400 G0 G90 X-260 Y-190 Z0          ;Definition of pocket profiles
     G1 X-200 Y30                   ;(External profile)
     X-200 Y210
     G2 G6 X-120 Y290 I-120 J210
     G1 X100 Y170
     G3 G6 X220 Y290 I100 J290
     G1 X360 Y290
     X360 Y-10
     G2 G6 X300 Y-70 I300 J-10
     G3 G6 X180 Y-190 I300 J-190
     G1 X-260 Y-190

     G0 X230 Y170                   ;First island profile definition
     G1 X290 Y170
     X230 Y50
     X150 Y90
     G3 G6 X230 Y170 I150 J170

     G0 X-120 Y90                   ;Second island profile definition
     G1 X20 Y90
     X20 Y-50
     X-120 Y-50
N500 X-120 Y90                      ;End of contour definition
```

Programming example, with automatic tool changer. The "x" of the figure indicate the initial points of each profile.




```
(TOR1=9, TOI1=0, TOL1=25,TOK1=0)     ;Tool 1 dimensions
(TOR2=3.6, TOI2=0, TOL2=20,TOK2=0)   ;Tool 2 dimensions
(TOR3=9, TOI3=0, TOL3=25,TOK3=0)     ;Tool 3 dimensions
G0 G17 G43 G90 X0 Y0 Z25 S800        ;Initial positioning
G66 D100 R200 F300 S400 E500         ;Irregular pocket description
M30                                  ;End of program

N100  G81 Z5 I-40 T3 D3 M6                          ;Definition of drilling operation
N200  G67 B10 C5 I-40 R5 K1 V100 F500 T1 D1 M6      ;Definition of roughing operation
N300  G68 B0 L0.5 Q1 V100 F300 T2 D2 M6             ;Definition of finishing operation

N400  G0 G90 X-300 Y50 Z3                           ;Definition of pocket profiles
      G1 Y190                                       ;(External profile)
      G2 G6 X-270 Y220 I-270 J190
      G1 X170
      X300 Y150
      Y50
      G3 G6 X300 Y-50 I300 J0
      G1 G36 R50 Y-220
      X-30
      G39 R50 X-100 Y-150
      X-170 Y-220
      X-270
      G2 G6 X-300 Y-190 I-270 J-190
      G1 Y-50
      X-240
      Y50
      X-300

      G0 X-120 Y80                                  ;First island contour definition
      G2 G6 X-80 Y80 I-100 J80                      ;(Contour a)
      G1 Y-80
      G2 G6 X-120 Y-80 I-100 J-80
      G1 Y80
```

```
      G0 X-40 Y0                        ;(Contour b)
      G2 G6 X-40 Y0 I-100 J0
      G0 X-180 Y20                      ;(Contour c)
      G1 X-20
      G2 G6 X-20 Y-20 I-20 J0
      G1 X-180
      G2 G6 X-180 Y20 I-180 J0

      G0 X150 Y140                      ;Second island profile definition
      G1 X170 Y110                      ;(Contour d)
      Y-110
      X150 Y-140
      X130 Y-110
      Y110
      X150 Y140
      G0 X110 Y0                        ;(Contour e)
N500  G2 G6 X110 Y0 I150 J0             ;End of contour definition
```

11.2 3D POCKETS

The cycle calling function G66 is not modal; therefore, it must be programmed every time a 3D pocket is to be executed.

A block containing function G66 may not contain any other function. Its format is:

G66 R I C J F K S E

R (0-9999) & I (0-9999) Label number of the first block (R) and last block (I) defining the roughing operation.
When not setting "I" only block "R" is executed.
When not setting "R" there is no roughing operation.

C (0-9999) & J (0-9999) Label number of the first block (C) and last block (J) defining the semi-finishing operation.
When not setting "J" only block "c" is executed.
When not setting "C" there is no semi-finishing operation.

F (0-9999) & K (0-9999) Label number of the first block (F) and last block (K) defining the finishing operation.
When not setting "K" only block "F" is executed.
When not setting "F" there is no finishing operation.

S (0-9999) & E (0-9999) Label number of the first block (S) and last block (E) defining the geometry of the profiles forming the pocket.
Both parameters must be set.

Programming example:

```
      G00 G90 X100 Y200 Z50 F5000 T1 D2  ;Initial positioning
      M06
      G66 R100 C200 J210 F300  S400 E500 ;Definition of irregular pocket canned cycle
      M30                                ;End of program

N100 G67 .....                           ;Defines the roughing operation
N200 .....                               ;Starts the semi-finishing operation
      G67 .....
N210 .....                               ;End the semi-finishing operation
N300 G68 .....                           ;Defines the finishing operation
N400 G0 G90 X300 Y50 Z3                  ;Starts the geometry description
      .....
      .....
N500 G2 G6 X300 Y50 I150 J0              ;End of geometry description
```

Basic operation:

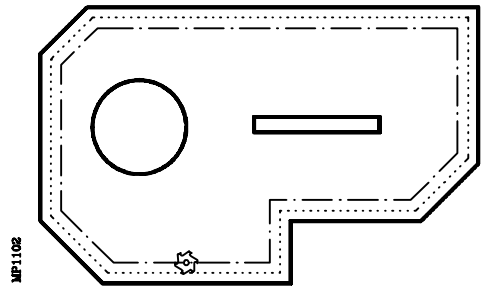
1.- Roughing operation. Only if it has been programmed.

It consists of several surface milling passes, until the total depth programmed has been reached. On each surface milling pass, the steps below will be followed depending on the type of machining that has been programmed:

Case A: When the machining paths are linear and maintain a certain angle with the abscissa axis.

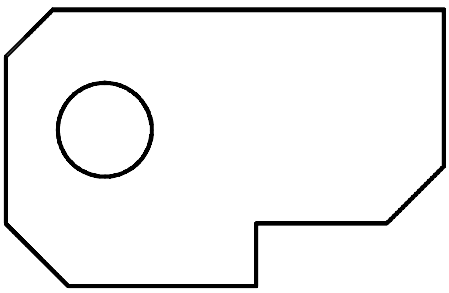
* It first contours the external profile of the part.

  If the finishing operation has been selected on the cycle call, this contouring is performed leaving the finishing stock programmed for the finishing pass.

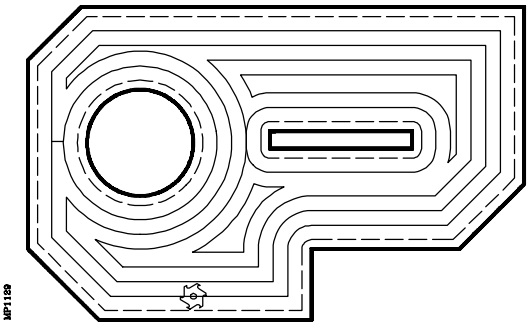


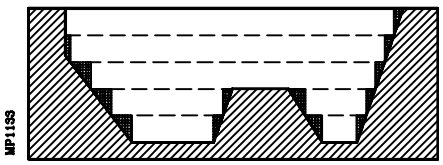
* Next the milling operation, with the programmed feed and steps.

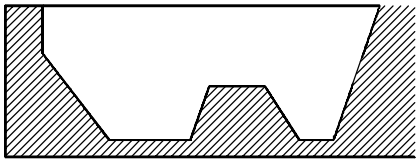
  If, while milling, an island is run into for the first time, it will be contoured.



After the contouring and the remaining times, the tool will pass over the island, withdrawing along the longitudinal axis, to the reference plane, and will continue machining once the island has been cleared.


MP1189


MP1193

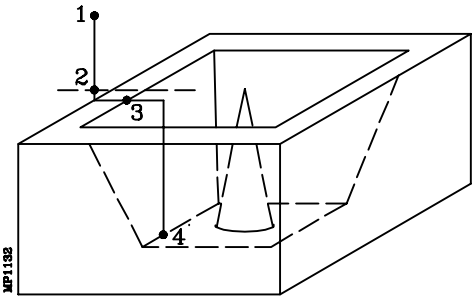

MP1194

After cycle conditions

Once the canned cycle has ended, the active feedrate will be the last one programmed. The one corresponding to the roughing or finishing operation. On the other hand, the CNC will assume functions G00, G40 and G90.

Reference coordinates

The irregular pocket canned cycle has four coordinates along the longitudinal axis, usually perpendicular to the plane (selected with G15), which, due to their importance, are described next:



1.- Starting plane coordinate. Given by the tool position at the beginning of the cycle.

2.- Reference plane coordinate. It must be programmed in absolute values and it represents a part approaching coordinate.

3.- Part surface coordinate (top). It is programmed in absolute values and in the first profile defining block.

4.- Machining depth coordinate (bottom. It must be programmed in absolute values.

11.2.1 ROUGHING OPERATION

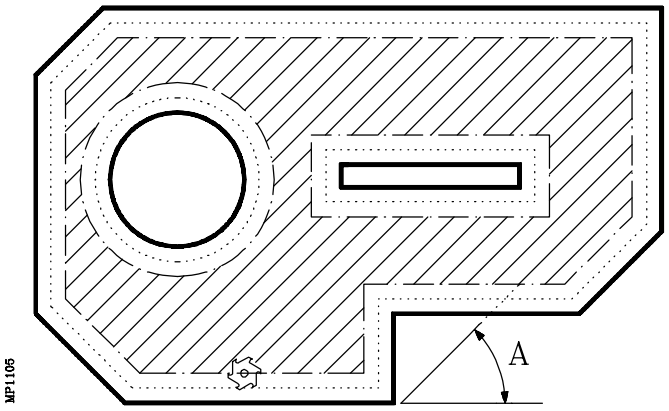
This is the main operation in the machining of an irregular pocket, and its programming is optional.

It will be programmed in a block which will need to bear a label number in order to indicate to the canned cycle the block where the roughing operation is defined.

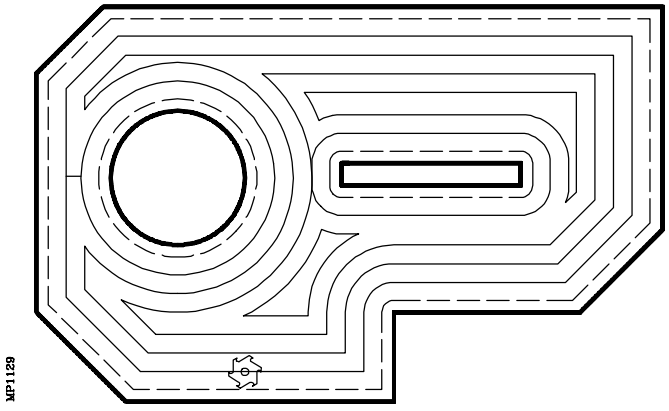
```
Example: G66 R100 C200 F300 S400 E500  ; Definition of the irregular pocket cycle.
         N100 G67 .....                ; Definition of the roughing operation.
```

The function for the roughing operation is G67 and it cannot be executed independently from the G66.
Its programming format: **G67 A B C I R V F S T D M**

A(+/-5.5) Defines the angle which forms the roughing path with the abscissa axis.



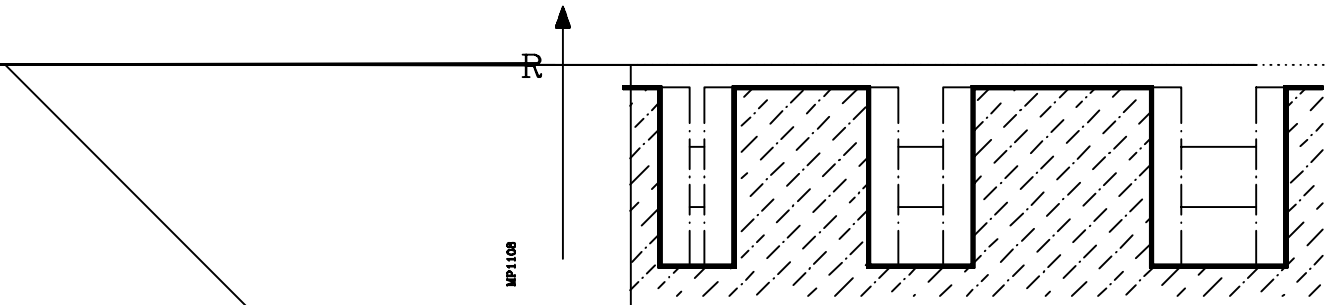
If parameter "A" is not programmed, the roughing operation is carried out following concentric paths. It will be machined as fast as possible since it does not have to go over the islands.



B(+/-5.5) Defines the machining pass along the longitudinal axis (depth of the roughing pass). It must be defined and it must have a value other than 0; otherwise, the roughing operation will be cancelled.

- If programmed with a positive sign, all the roughing will be performed

R(+/-5.5) Defines the reference plane coordinate and is programmed in absolute coordinates. It must be programmed.



V (5.5) Defines the tool penetrating feedrate.

If not programmed or programmed with a value of "0", the CNC will assume 50% of the feedrate in the plane (F).

F (5.5) Optional. Defines the machining feedrate in the plane.

S (5.5) Optional. Defines the spindle speed.

T (4) Defines the tool used for the roughing operation. It must be programmed.

D (4) Optional. Defines the tool offset number.

M Optional. Up to 7 miscellaneous M functions can be programmed.

This operation allows M06 with an associated subroutine to be defined, and the tool change is performed before beginning the roughing operation.

11.2.2 SEMI-FINISHING OPERATION

This operation is optional.

It will be programmed in a block which will need to bear a label number in order to indicate to the canned cycle the block where the roughing operation is defined.

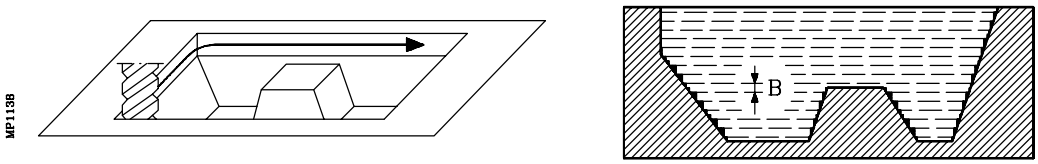
Example: G66 R100 **C200** F300 S400 E500 ; Definition of the irregular pocket cycle.
N200 G67 ..... ; Definition of the semi-finish operation.

The function for the semi-finishing operation is G67 and it cannot be executed independently from the G66.

Both the roughing and the semi-finishing operations are defined with G67; but, in different blocks. It is function G66 who indicates which is which by means of parameters "R" and "C".

Its programming format is: **G67 B I R V F S T D M**

B (±5.5) Defines the machining step along the longitudinal axis (semi-finishing pass). It must be programmed and with a value other than "0". Otherwise, the semi-finishing operation will be canceled.



- If programmed with a positive sign, the whole semi-finish operation will be carried out with the same machining pass and the canned cycle will calculate a pass equal or smaller than the one programmed.

- If programmed with a negative sign, the whole semi-finish operation will be run with the programmed pass. The canned cycle will adjust the last pass to obtain the total programmed depth.

I (±5.5) Defines the total pocket depth and it is programmed in absolute coordinates.

If there is a roughing operation and it is not programmed, the CNC takes the value defined for the roughing operation.

If there is no roughing operation, it must be programmed.

R (±5.5) Defines the coordinate of the reference plane and it is programmed in absolute values.

If there is a roughing operation and it is not programmed, the CNC takes the value defined for the roughing operation.

If there is no roughing operation, it must be programmed.

V (5.5) Defines the tool penetrating feedrate.

If not programmed or programmed with a value of "0", the CNC will assume 50% of the feedrate in the plane (F).

F (5.5) Optional. Defines the machining feedrate in the plane.

S (5.5) Optional. Defines the spindle speed.

T (4) Defines the tool used for the semi-finishing operation. It must be programmed.

D (4) Optional. Defines the tool offset number.

M Optional. Up to 7 miscellaneous M functions can be programmed.

This operation allows M06 with an associated subroutine to be defined, and the tool change is performed before beginning the semi-finishing operation.

11.2.3 FINISHING OPERATION

This operation is optional.

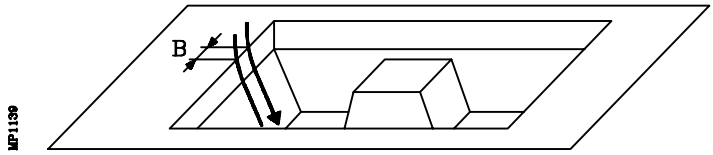
It will be programmed in a block which will need to bear a label number in order to indicate to the canned cycle the block where the roughing operation is defined.

Example: G66 R100 C200 **F300** S400 E500 ; Definition of the irregular pocket cycle.
N300 G67 ..... ; Definition of the finishing operation.

The function for the finishing operation is G68 and it cannot be executed independently from the G66.

Its programming format is: **G68 B L Q J I R V F S T D M**

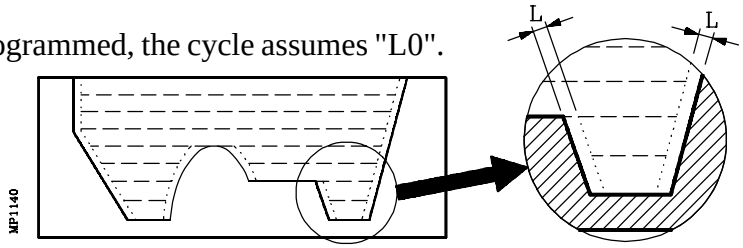
B (5.5) Defines the pass in the plane between two 3D paths of the finishing operation. It must be defined and with a value other than "0".



L (±5.5) Defines the value of the finishing stock on the side walls of the pocket left by the roughing and semi-finishing operations.

There is no finishing stock left on top of the islands nor on the bottom of the pocket.

If not programmed, the cycle assumes "L0".



Q Indicates the direction of the finishing pass.

Q = 1 All the passes will be inward from the top of the pocket to its bottom

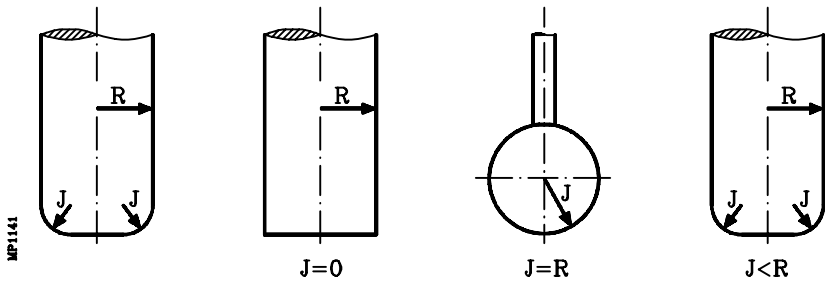
Q = 2 All the passes will be outward from the bottom of the pocket to the top.

Q = 0 Alternating direction for every 2 consecutive paths.

Any other value will generate the corresponding error. If parameter "Q" is not programmed, the cycle assumes "Q0".

J (5.5) Indicates the tool tip radius and, therefore, the type of finishing tool being used.

Depending on the radius assigned to the tool in the tool offset table (of the CNC variables: "TOR" + "TOI") and the value of assigned to this parameter, three tool types may be defined.



FLAT ..... If J is not programmed or J = 0
BALL-END ..... If J = R
TORIC (Corner rounding).... If J <> 0 (other than 0) and J < R

I (±5.5) Defines the total pocket depth and it is given in absolute coordinates.

- If defined, the cycle will take it into account during the finishing operation.
- If not defined and the pocket has a roughing operation, the cycle will assume the value defined for the roughing operation.
- If not defined and the pocket has no roughing operation, but it has a semi-finishing operation, the cycle will assume the one define in the semi-finishing operation.
- If the pocket has neither roughing nor semi-finishing operation, this parameter must be defined.

R (±5.5) Defines the coordinate of the reference plane and it must be given in absolute values.

- If defined, the cycle will take it into account during the finishing operation.
- If not defined and the pocket has a roughing operation, the cycle will assume the value defined for the roughing operation.
- If not defined and the pocket has no roughing operation, but it has a semi-finishing operation, the cycle will assume the one define in the semi-finishing operation.
- If the pocket has neither roughing nor semi-finishing operation, this parameter must be defined.

V (5.5) Defines the tool penetrating feedrate.

If not programmed or programmed with a value of "0", the CNC will assume 50% of the feedrate in the plane (F).

F (5.5) Optional. Defines the machining feedrate in the plane.

S (5.5) Optional. Defines the spindle speed.

T (4) Defines the tool used for the finishing operation. It must be programmed.

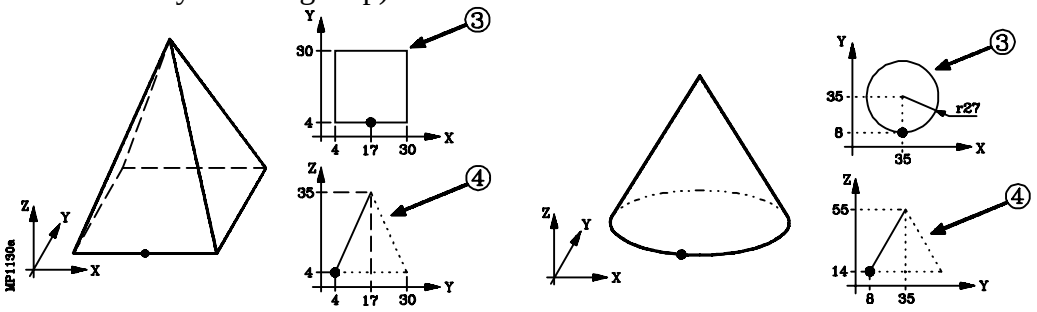
D (4) Optional. Defines the tool offset number.

M Optional. Up to 7 miscellaneous M functions can be programmed.

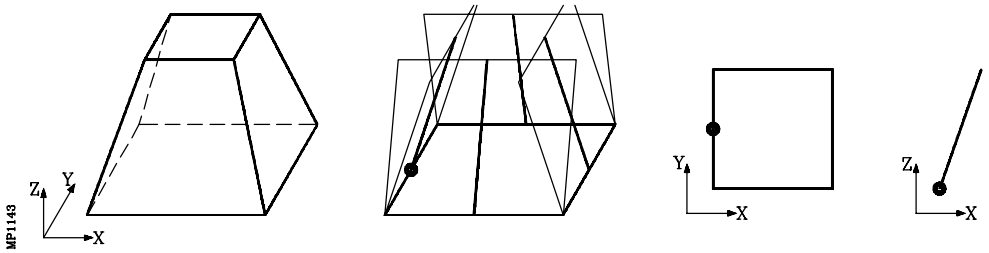
This operation allows M06 with an associated subroutine to be defined, and the tool change is performed before beginning the finishing operation.

11.2.4 PROFILE OR CONTOUR GEOMETRY

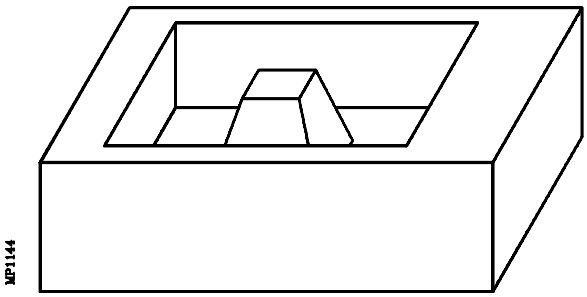
To define the contours or profiles of a 3D pocket, one must specify the plane profile or horizontal cross section (3) and the depth profile or vertical cross section (4) of all contours (even when they are straight up).



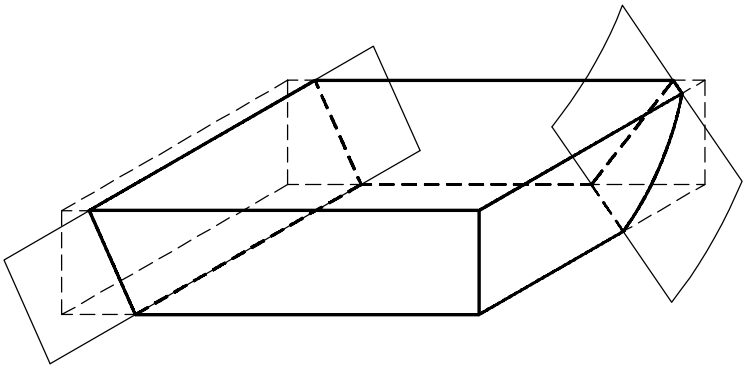
Since the canned cycle applies the same depth profile to the whole contour, the same start point must be used to define the plane profile as for the depth profile.



Example of a 3D pocket:



3D contours with more than one depth profile are also possible. These contours are called "composite 3D profiles" and will be described later on.



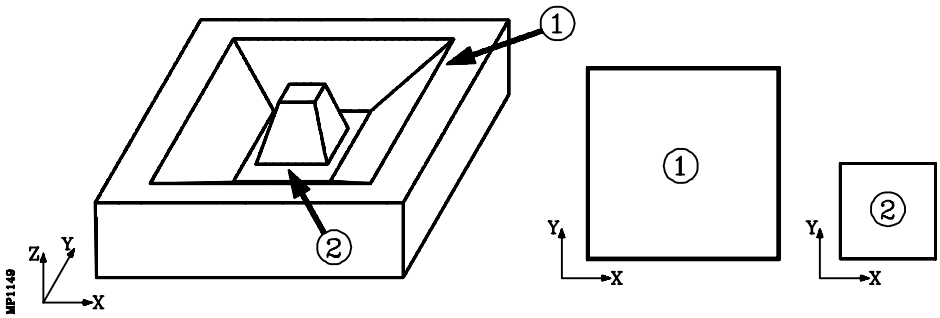
11.2.5 PROFILE PROGRAMMING RULES

When programming inside or outside contours of an irregular 3D pocket (with islands) , the following rules must be complied with:

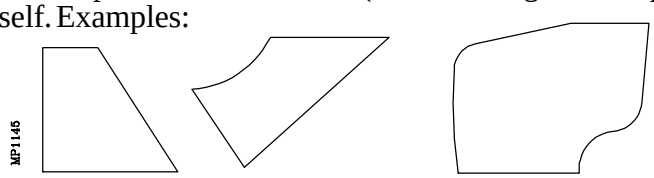
1.- The profile in the main plane indicates the shape of the contour.

Since a 3D contour has an infinite number of different profiles (1 per each depth coordinate), the following must be programmed:

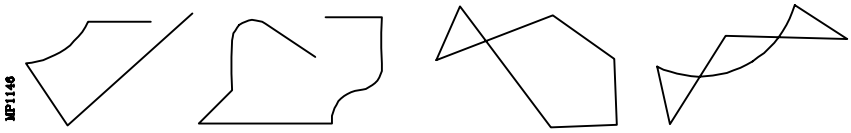
* For the outside contour of the pocket: the one corresponding to the surface coordinate or top of the part (1).
* For the inside contours: the one corresponding to the base or bottom (2).



2.- The profile in the plane must be closed (same starting and end points) and it must not intersect itself. Examples:



The following examples cause a geometry error:



3.- The depth profile (vertical cross section) must be programmed with any of the axes of the active plane. If the active plane is the XY and the perpendicular axis is the Z axis, one must program: G16XZ or G16YZ.

All profiles, plane and depth, must start with the definition of the plane containing it.

Example: G16 XY ..... Beginning of the outside profile definition
----- *plane profile definition* -----
G16 XZ
------ *depth profile definition* ----

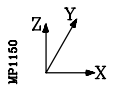
G16 XY ..... Beginning of the island definition
----- *plane profile definition* -----
G16 XZ
------ *depth profile definition* ----

4.- The depth profile must be defined after having defined the plane profile.

The beginning points of the plane profile and depth profile must be the same one.

Nevertheless, the depth profile must be programmed:

* For the outside contour of the pocket starting from the top or surface coordinate (1).
* For the inside contours, islands, starting from the bottom or base coordinate (2).



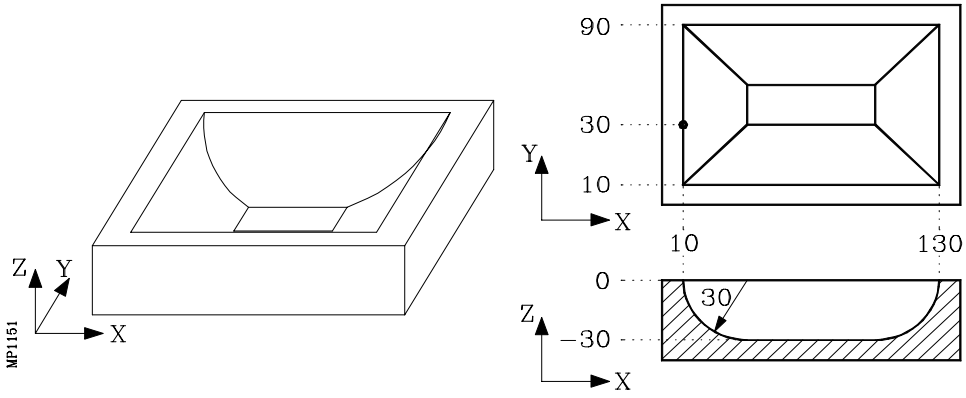
5.- The depth profile must be open and without direction changes along its path. In other words, it cannot zig-zag.

Examples:

The following examples cause geometry errors.

11.2.5.1 PROGRAMMING EXAMPLES

Example of a pocket without islands:

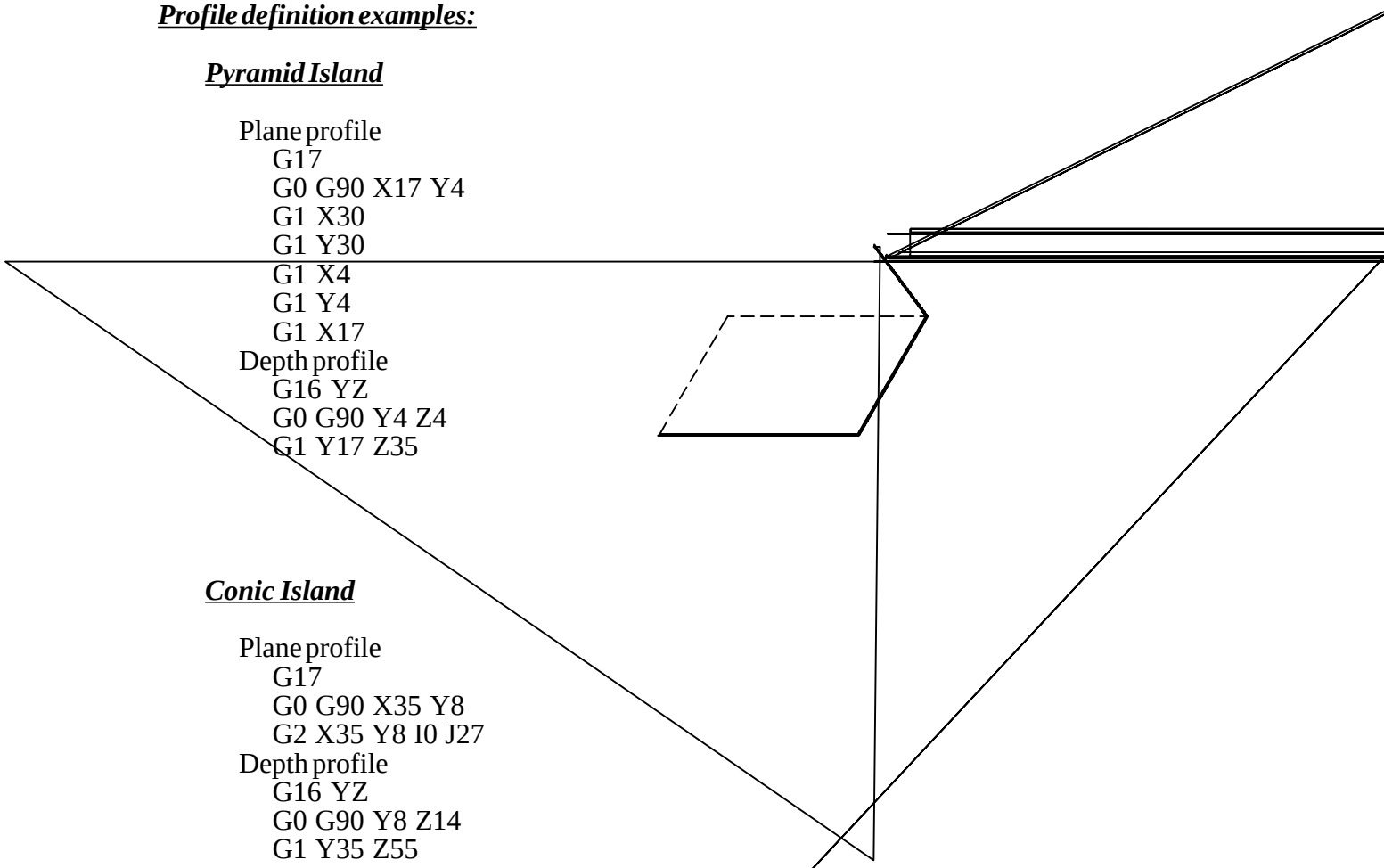



```
(TOR1=2.5,TOL1=20,TOI1=0,TOK1=0)
G17 G0 G43 G90 Z50 S1000 M4
G5
G66 R200 C250 F300 S400 E500..... ;3D pocket definition
M30

N200  G67 B5 C4 I-30 R5 V100 F400 T1D1 M6 ..... ;Roughing  operation
N250  G67 B2 I-30 R5 V100 F550 T2D1 M6..... ;Semi-finishing  operation
N300  G68 B1.5 L0.75 Q0 I-30 R5 V80 F275 T3D1 M6..... ;Finishing  operation

N400  G17..... ;Beginning of the pocket geometry definition
      G90 G0 X10 Y30 Z0..... ;Plane profile (horizontal cross section)
      G1  Y90
          X130
          Y10
          X10
          Y30
      G16 XZ ..... ;Depth profile (vertical cross section)
      G0  X10 Z0
N500  G3  X40 Z-30 I30 K0 ..... ;End of the pocket geometry definition
```

Profile definition examples:

Pyramid Island

Plane profile
G17
G0 G90 X17 Y4
G1 X30
G1 Y30
G1 X4
G1 Y4
G1 X17
Depth profile
G16 YZ
G0 G90 Y4 Z4
G1 Y17 Z35



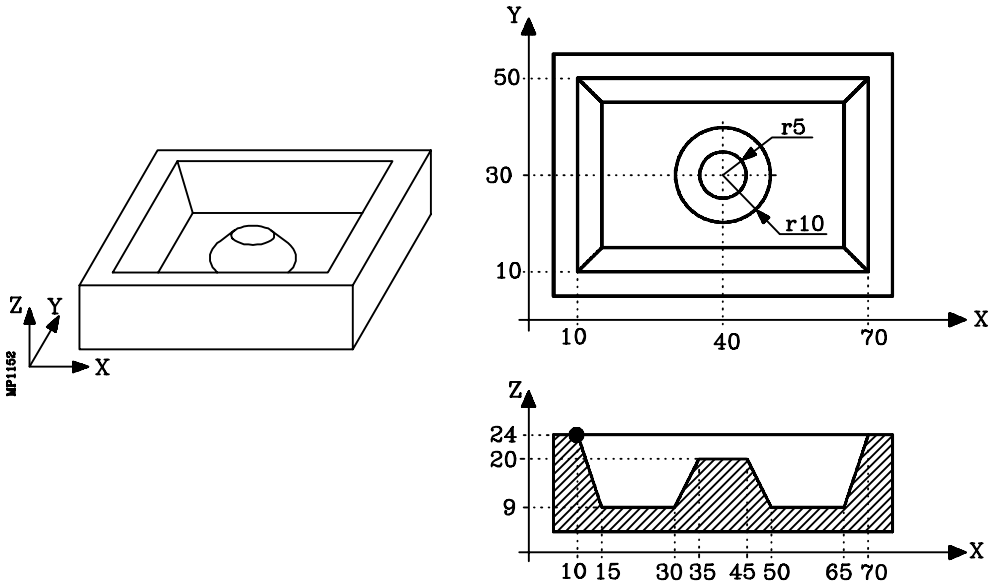
Conic Island

Plane profile
G17
G0 G90 X35 Y8
G2 X35 Y8 I0 J27
Depth profile
G16 YZ
G0 G90 Y8 Z14
G1 Y35 Z55

Semi-spherical Island

Plane profile
G17
G0 G90 X35 Y8
G2 X35 Y8 I0 J27
Depth profile
G16 YZ
G0 G90 Y8 Z14
G2 Y35 Z41 R27**3D POCKETS(EXAMPLES)**

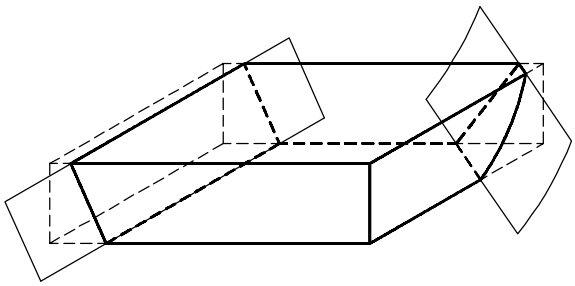
Example of a 3D pocket with islands:




```
(TOR1=2.5,TOL1=20,TOI1=0,TOK1=0)
G17 G0 G43 G90 Z50 S1000 M4
G5
G66 R200 C250 F300 S400 E500 ..... ;3D pocket definition
M30

N200  G67 B5 C4 I9 R25 V100 F400 T1D1 M6 ..... ;Roughing  operation
N250  G67 B2 I9 R25 V100 F550 T2D1 M6 ..... ;Semi-finishing  operation
N300  G68 B1.5 L0.75 Q0 I9 R25 V50 F275 T3D1 M6 ..... ;Finishing  operation

N400  G17 ..... ;Beginning of the pocket geometry definition
      G90 G0 X10 Y30 Z24 ..... ;Outside contour (plane profile)
      G1   Y50
           X70
           Y10
           X10
           Y30
      G16 XZ ..... ;Depth profile
      G0   X10 Z24
      G1   X15 Z9

      G17  ..... ;Island  definition
      G90 G0 X30 Y30 ..... ;Plane profile
      G2   X30 Y30 I10 K0
      G16 XZ ..... ;Depth profile
      G90 G0 X30 Z9
N500  G1   X35 Z20 ..... ;End of the pocket geometry definition
```

11.2.6 COMPOSITE 3D PROFILES

A composite 3D profile is a 3D contour with more than one depth profile.

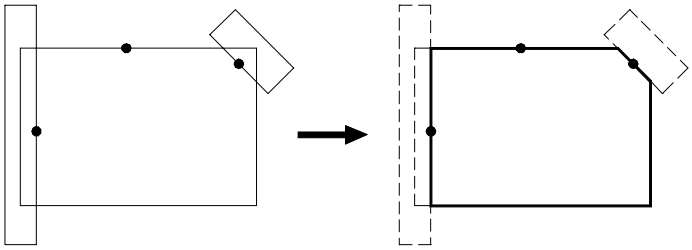


It is defined by means of the intersection of several contours with different depth profiles.

Each contour is defined by a profile in the plane and a depth profile. All the contours must meet the following conditions:

- The plane profile must contain the corresponding sides completely.
- Only a depth profile per contour must be defined.
- The plane profile and the depth profile of the contour gathering several sides must start at the same point.

The resulting plane profile will be formed by the intersection of the plane profiles of each element or contour.



Each wall of the resulting profile will assume the corresponding depth profile.

11.2.6.1 PROFILE INTESECTING RULES

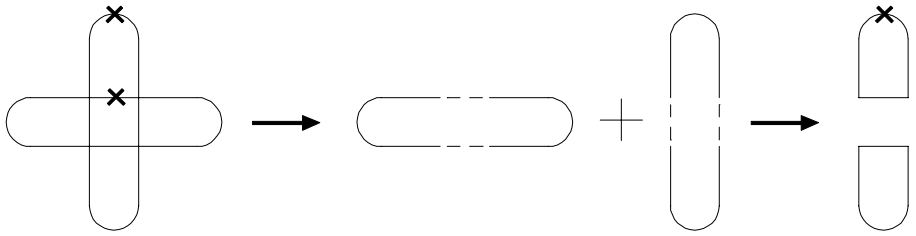
The plane profile intersecting rules are:

1.- At a profile intersection, each contour is divided into several lines which could be grouped as:

- Lines external to the other contour.
- Lines internal to the other contour.

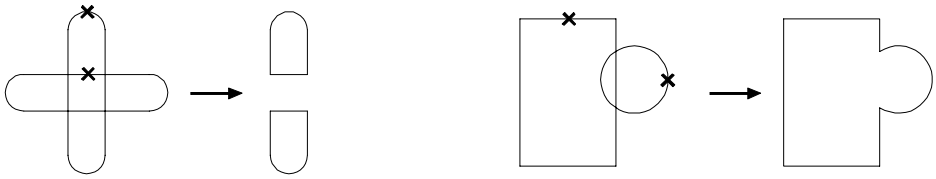
The starting point of each contour (x) determines the group of lines to be selected.

The following example shows the selection process using a solid line for the lines external to the other contour and a dotted line the internal ones.

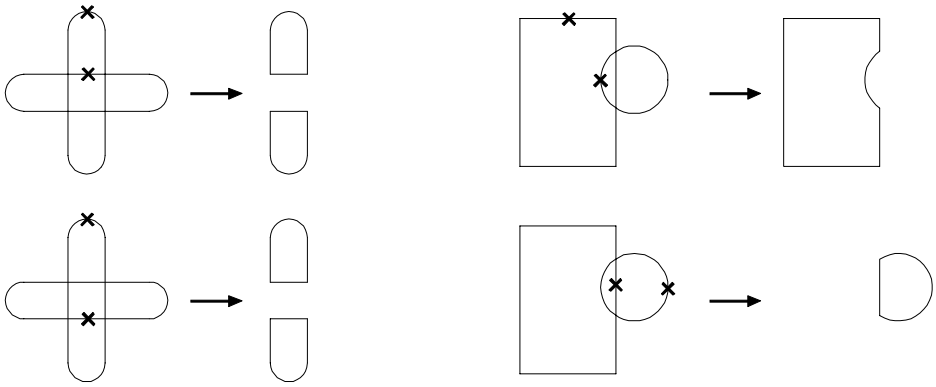


Profile intersection examples:

Boolean addition



Boolean subtraction



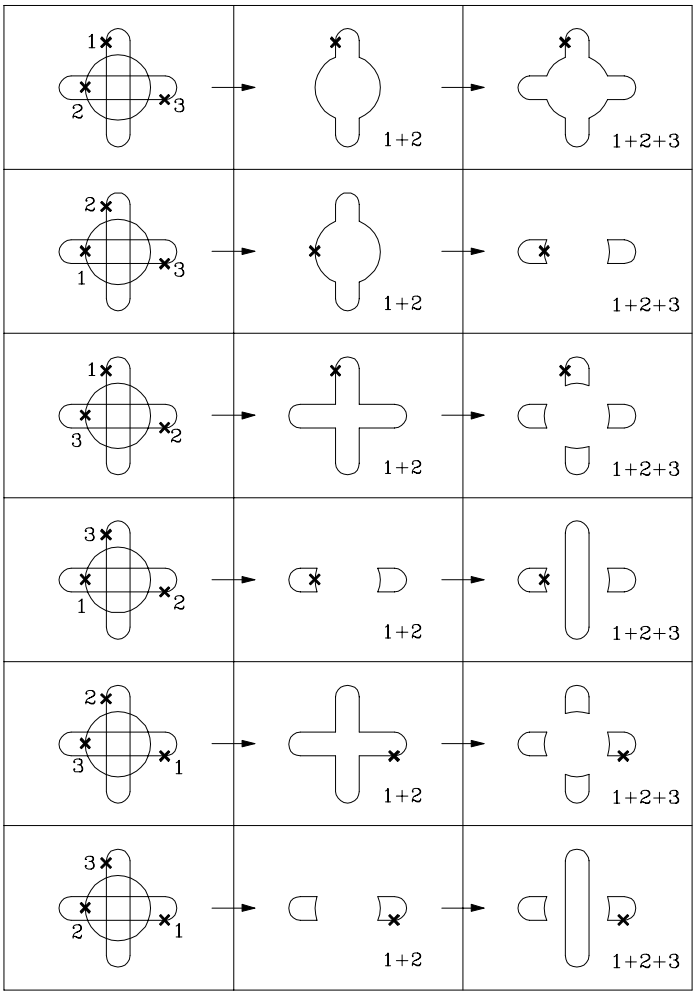
Boolean intersection

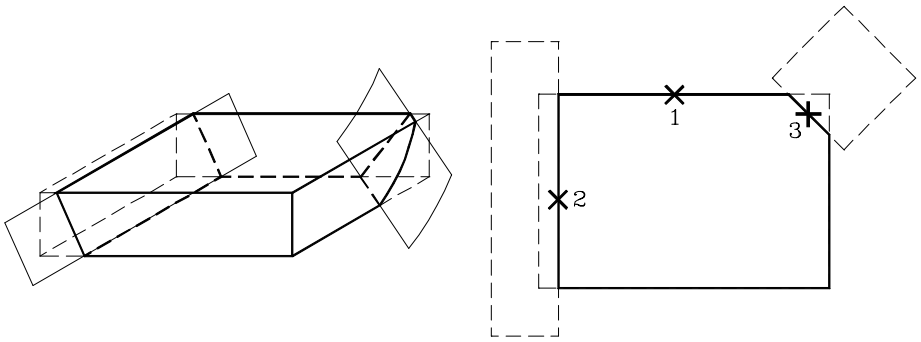
2.- The programming order of the various profiles is a determining factor when caring out an intersection of 3 or more profiles.

The profile intersecting process is done according the order (sequence) followed when programming the profiles. This way, after doing the intersection of the two profiles programmed first, the resulting profile will be intersected with the third one and so on.

The starting point of the resulting profiles always coincides with the starting point used to define the first profile.

Examples:

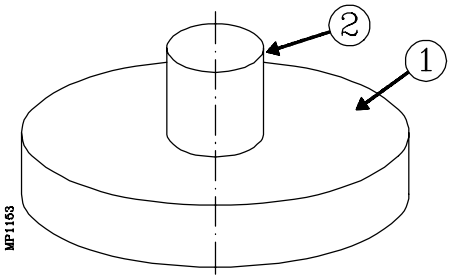




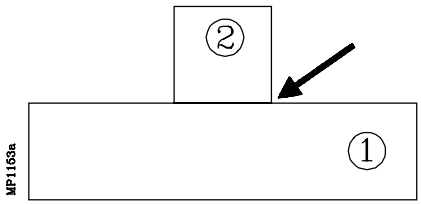
11.2.7 STACKED PROFILES

When 2 or more profiles stack on top of each other) the following considerations must be taken into account.

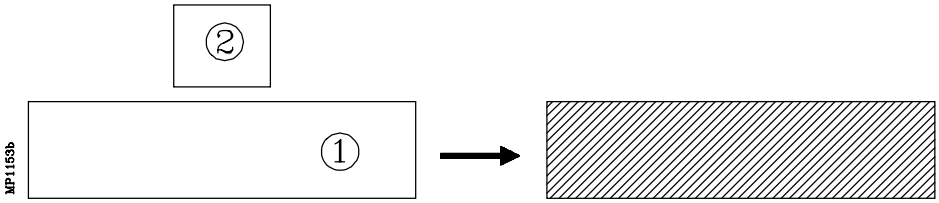
For clarity sakes, refer to the drawing on the right which consists of 2 stacked profiles: 1 and 2.



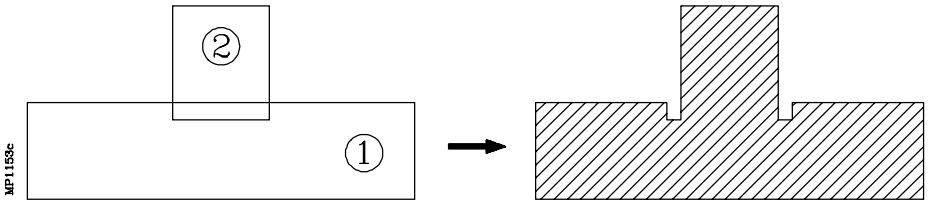
The base coordinate of the top profile (2) must coincide with the surface coordinate of the bottom profile (1).



If there is a gap between them, the cycle will consider that they are 2 different profiles and it will eliminate the top profile when executing the bottom one.



If the profiles mix, the canned cycle will make a groove around the top profile when running the finishing pass.



11.2.8 PROFILE PROGRAMMING SYNTAX

The outside profile and the inside profiles or islands which are programmed must be defined by simple geometrical elements such as straight lines or arcs.

The first definition block (where the external profile starts) and the last (where the last profile defined ends) must be provided with the block label number. These label numbers will be those which indicate to the canned cycle the beginning and end of the geometry description of the profiles which make up the pocket.

```
Example:  G66 R100 C200 F300 S400 E500   ;Irregular pocket canned cycle definition
          N400 G17                       ;Beginning of geometry description
          ------ ----- ---- ---
          N500 G2 Y50 Z-15 I10 K0        ;End of geometry description
```

The profile programming syntax must comply with the following rules:

1.- The first profile defining block must have a label number to indicate to the G66 canned cycle the beginning of the geometry description.

2.- First, the outside pocket contour must be defined and, then, the contour of each island.

3.- When a contour has more than one depth profile, the contours must be defined one by one indicating, on each one, the plane profile and, then, its depth profile.

4.- The first profile defining block of the plane profile as well as that of the depth profile must contain function G00 (indicative of the beginning of the profile).

Care must be taken to program G01, G02 or G03 in the block following the definition of the beginning, as G00 is modal, thus preventing the CNC from interpreting the following blocks as the beginnings of a new profile.

5.- The last profile defining block must have a label number to indicate to the G66 canned cycle the end of the geometry description.

Example:

```
    G66 R200 C250 F300 S400 E500 ..... ;3D pocket definition

N400  G17 ..... ;Beginning of the pocket geometry description
    G0  G90 X5 Y-26 Z0 ..... ;Outside contour (plane profile)
    --- ---- ---- ----
    --- ---- ---- ----
    G16 XZ ..... ;Depth profile
    G0
    --- ---- ---- ----
    --- ---- ---- ----
    G17 ..... ;Island
    G0 X30 Y-6 ..... ;Plane profile
    --- ---- ---- ----
    --- ---- ---- ----
    G16 XZ ..... ;Depth profile
    G0
    --- ---- ---- ----
    --- ---- ---- ----
N500  G3 Y-21 Z0 J-5 K0 ..... ;End of the pocket geometry description
```

6.- Profiles are described as programmed paths, it being possible to include corner rounding, chamfers, etc., following the syntax rules defined for this purpose.

7.- Mirror images, scaling factor changes, rotation of coordinate system, zero offsets, etc., cannot be programmed in the description of profiles.

8.- Nor is it possible to program blocks in high level language, such as jumps, subroutine calls or parametric programming.

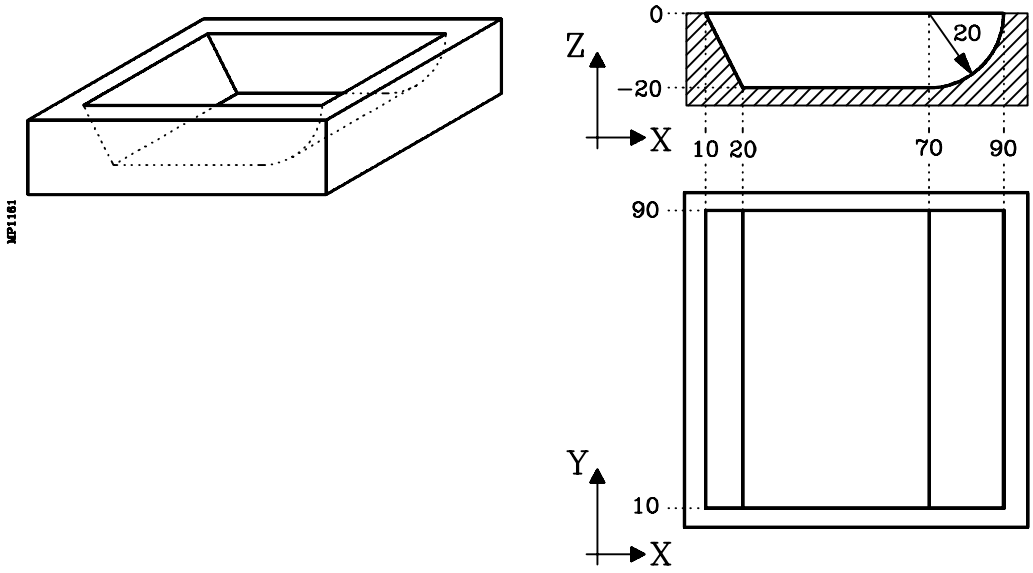
9.- Other canned cycles cannot be programmed.

In addition to the G00 function, which has a special meaning, the irregular pocket canned cycle allows the use of the following functions for the definition of profiles.

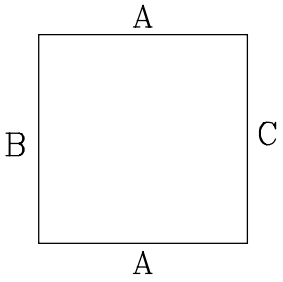
G01 Linear interpolation
G02 Clockwise circular interpolation
G03 Counter-clockwise circular interpolation
G06 Arc center in absolute coordinates
G08 Arc tangent to previous path.
G09 Arc defined by three points
G16 Main plane section by two directions
G17 Main plane X-Y and longitudinal Z (perpendicular)
G18 Main plane Z-X and longitudinal Y (perpendicular)
G19 Main plane Y-Z and longitudinal X (perpendicular)
G36 Automatic radius blend (controlled corner rounding)
G39 Chamfer
G53 Programming with respect to machine reference zero (home)
G70 Programming in inches
G71 Programming in millimeters
G90 Absolute programming
G91 Incremental programming
G93 Polar origin preset

11.2.9 EXAMPLES

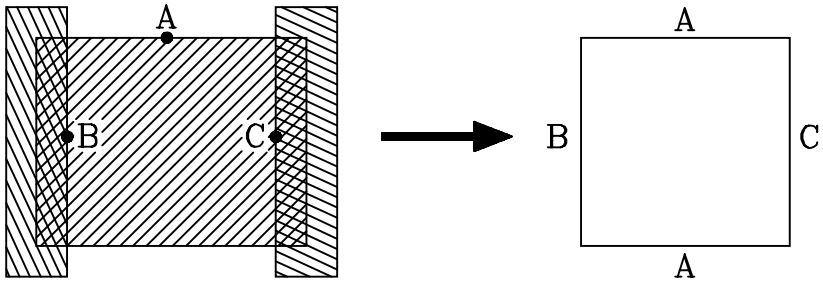
Example 1, Pocket without islands:



In this example, the island has 3 types of depth profiles: A, B and C.

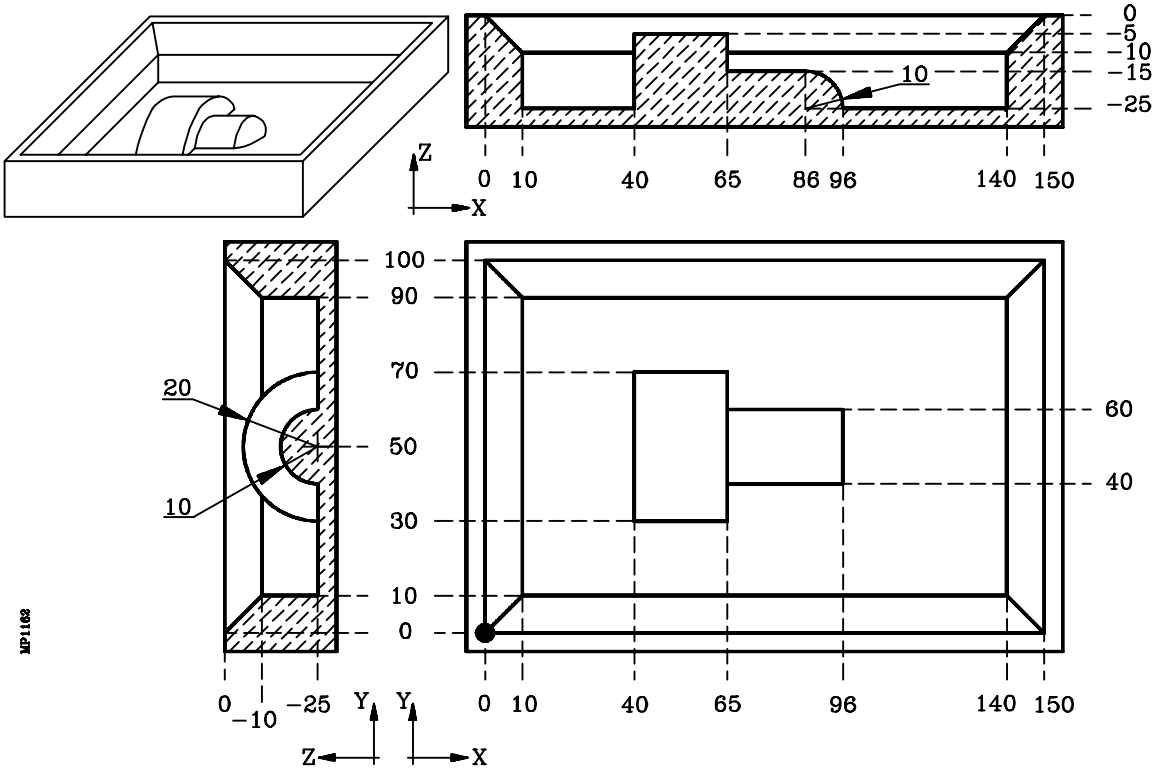


3 contours are used to define the island: A-type contour, B-type contour and C-type contour.

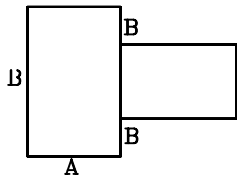


```
        (TOR1=2.5,TOL1=20,TOI1=0,TOK1=0)
        G17 G0 G43 G90 Z50 S1000 M4
        G5
        G66 R200 C250 F300 S400 E500 ..... ;3D pocket definition
        M30

N200    G67 B5 C4 I-20 R5 V100 F400 T1D1 M6 ..... ;Roughing operation
N250    G67 B2 I-20 R5 V100 F550 T2D1 M6 ..... ;Semi-finishing operation
N300    G68 B1.5 L0.75 Q0 I-20 R5 V80 F275 T3D1 M6 ..... ;Finishing operation


N400    G17 ..... ;Beginning of pocket geometry definition
        G0 G90 X50 Y90 Z0 ..... ;A-type contour (Plane profile)
        G1 X0
           Y10
           X100
           Y90
           X50
        G16 YZ ..... ;Depth profile
        G0 G90 Y90 Z0
        G1 Z-20


        G17 ..... ;B-type contour
        G0 G90 X10 Y50 ..... ;Plane profile
        G1 Y100
           X-10
           Y0
           X10
           Y50
        G16 XZ ..... ;Depth profile
        G0 G90 X10 Z0
        G1 X20 Z-20


        G17 ..... ;C-type contour
        G0 G90 X90 Y50 ..... ;Plane profile
        G1 Y100
           X110
           Y0
           X90
           Y50
        G16 XZ ..... ;Depth profile
        G0 G90 X90 Z0
N500    G2 X70 Z-20 I-20 K0 ..... ;End of pocket geometry definition
```

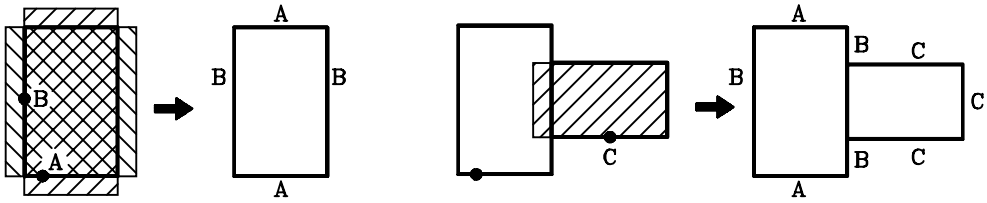
Example 2:



In this example, the island has 3 types of depth profiles: A, B and C.



3 contours are used to define the island: A-type contour, B-type contour and C-type contour.

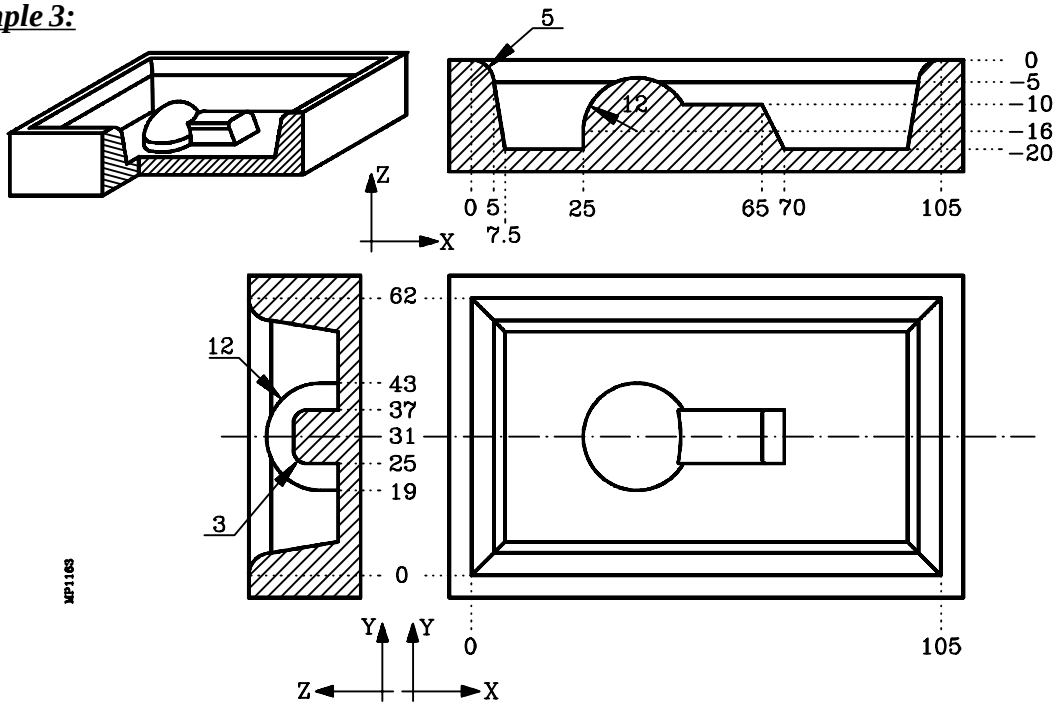


(TOR1=7.5,TOI1=0,TOR2=5,TOI2=0,TOR3=2.5,TOI3=0)
G17 G0 G43 G90 Z50 S1000 M4
G5
G66 R200 C250 F300 S400 E500 ..... ;3D pocket definition
M30

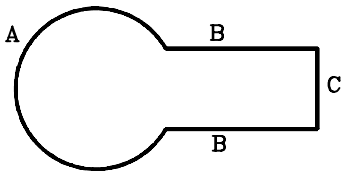
N200 G67 B7 C14 I-25 R3 V100 F500 T1D1 M6 ..... ;Roughing operation
N250 G67 B3 I-25 R3 V100 F625 T2D2 M6 ..... ;Semi-finishing operation
N300 G68 B1 L1 Q0 J0 I-25 R3 V100 F350 T3D3 M6 ..... ;Finishing operation**3D**

```
N400  G17 ..... ;Beginning of pocket geometry definition
      G0 G90 X0 Y0 Z0 ..... ;Outside contour (plane profile)
      G1 X150
         Y100
         X0
         Y0
      G16 XZ ..... ;Depth profile
      G0 G90 X0 Z0
      G1 X10 Z-10
         Z-25


      G17 ..... ;A-type profile
      G0 G90 X50 Y30 ..... ;Plane profile
      G1 X70
         Y70
         X35
         Y30
         X50
      G16 YZ ..... ;Depth profile
      G0 G90 Y30 Z-25
      G2 Y50 Z-5 J20 K0


      G17 ..... ;B-type profile
      G0 G90 X40 Y50 ..... ;Plane profile
      G1 Y25
         X65
         Y75
         X40
         Y50
      G16 XZ ..... ;Depth profile
      G0 G90 X40 Z-25
      G1 Z-5


      G17 ..... ;C-type profile
      G0 G90 X80 Y40 ..... ;Plane profile
      G1 X96
         Y60
         X60
         Y40
         X80
      G16 YZ ..... ;Depth profile
      G0 G90 Y40 Z-25
N500  G2 Y50 Z-15 J10 K0 ..... ;End of pocket geometry definition
```

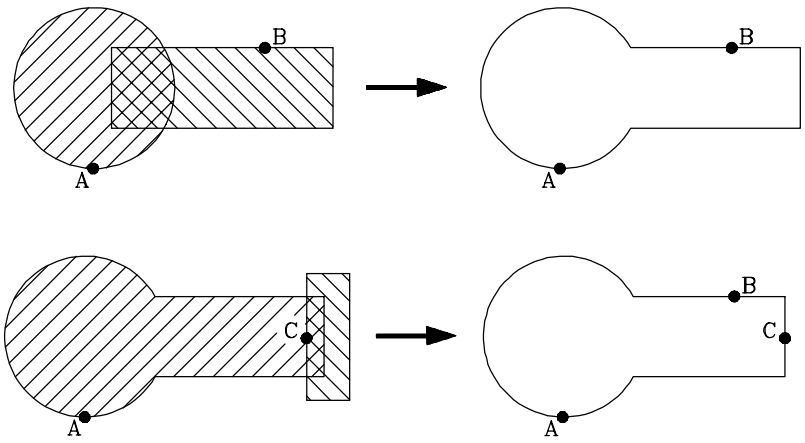
Example 3:



In this example, the island has 3 types of depth profiles: A, B and C.

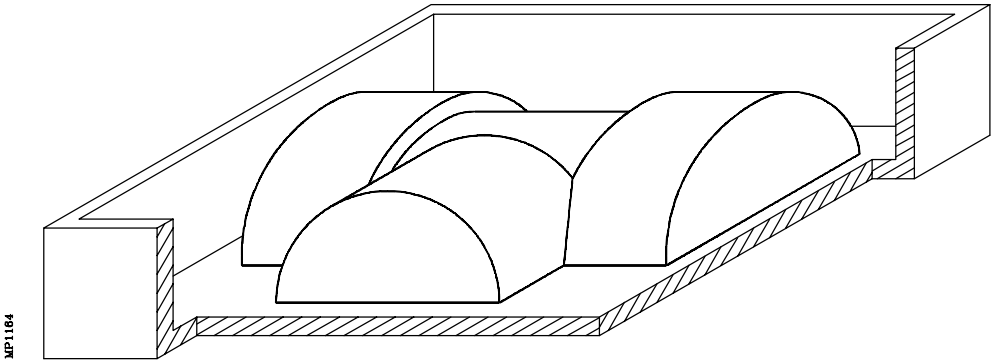


3 contours are used to define the island: A-type contour, B-type contour and C-type contour.

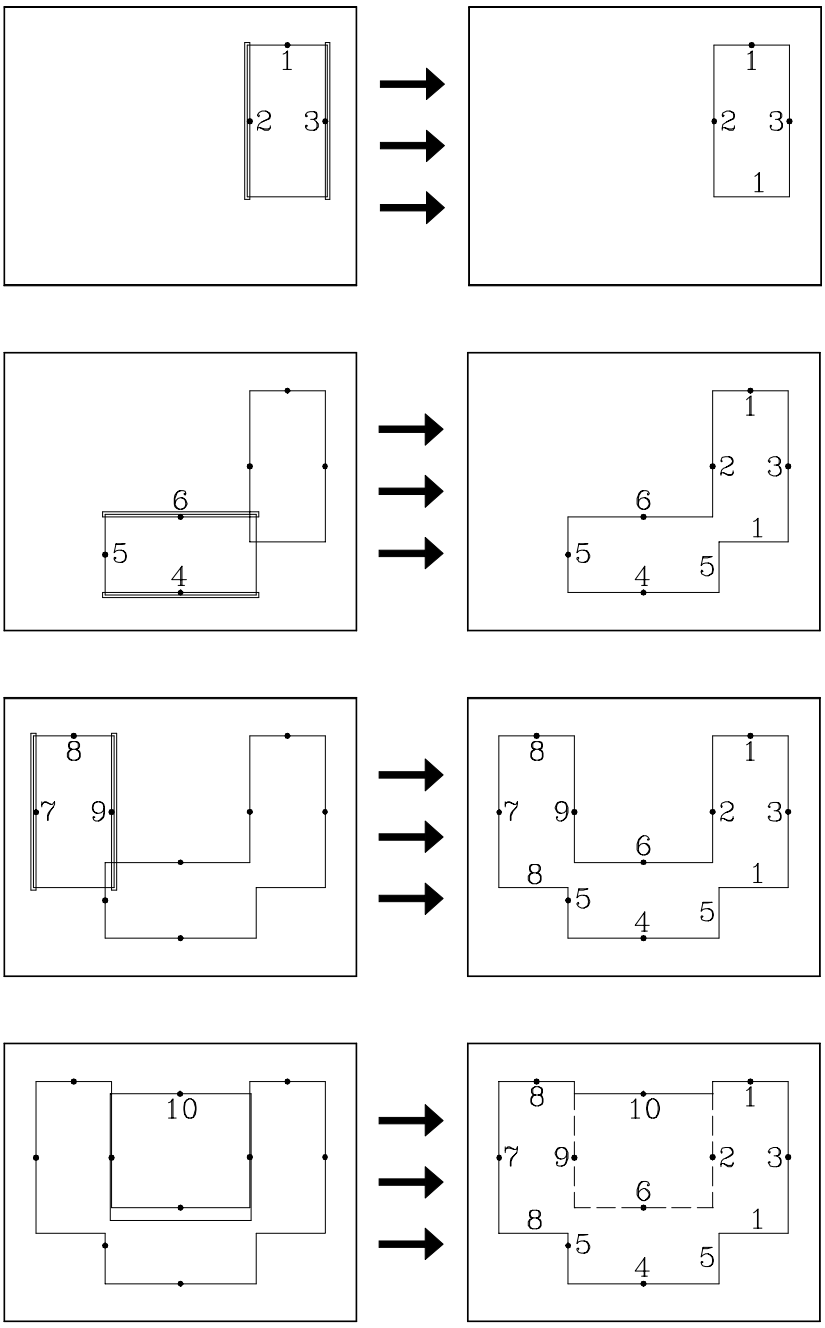


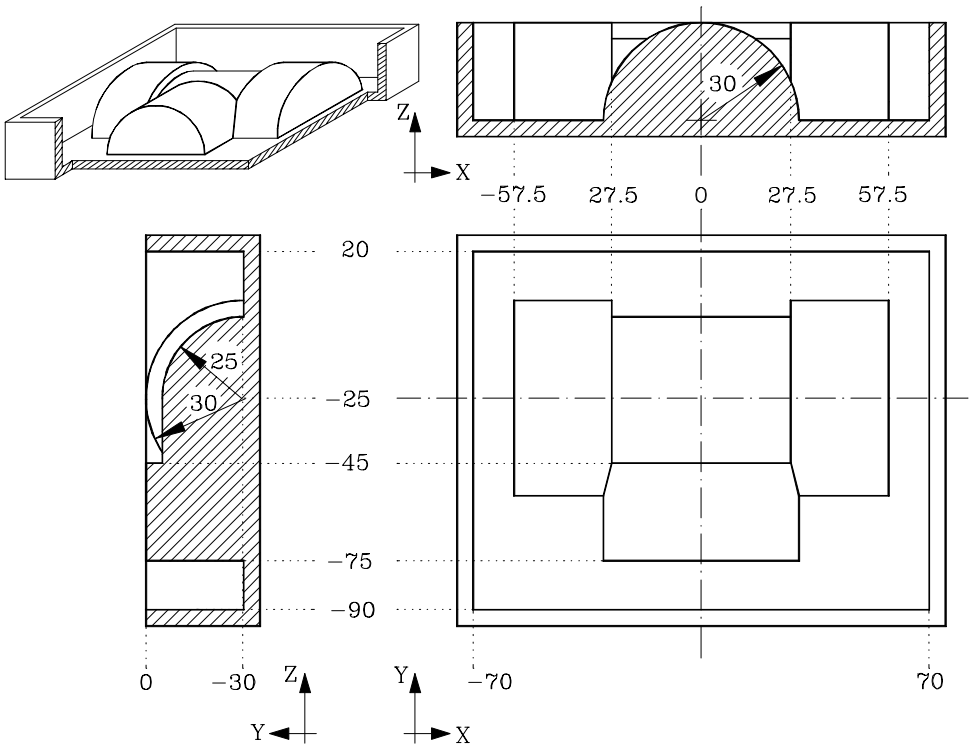
```
        (TOR1=4,TOI1=0,TOR2=2.5,TOI2=0)
        G17 G0 G43 G90 Z25 S1000 M3
        G66 R200 C250 F300 S400 E500 ..... ;3D pocket definition
        M30

N200  G67 B5 C4 I-20 R5 V100 F700 T1D1 M6 ..... ;Roughing  operation
N250  G67 B2 I-20 R5 V100 F850 T1D1 M6 ..... ;Semi-finishing operation
N300  G68 B1.5 L0.25 Q0 I-20 R5 V100 F500 T2D2 M6 ..... ;Finishing  operation
;

N400  G17 ..... ;Beginning of pocket geometry definition
        G0 G90 X0 Y0 Z0 ..... ;Outside contour (plane profile)
        G1 X105
            Y62
            X0
            Y0
        G16 XZ ..... ;Depth profile
        G0 X0 Z0
        G2 X5 Z-5 I0 K-5
        G1 X7.5 Z-20


        G17 ..... ;A-type contour
        G90 G0 X37 Y19..... ;Plane profile
        G2 I0 J12
        G16 YZ ..... ;Depth profile
        G0 Y19 Z-20
        G1 Z-16
        G2 Y31 Z-4 R12 ..... ;End of pocket geometry definition


        G17 ..... ;B-type contour
        G90 G0 X60 Y37..... ;Plane profile
        G1 X75
            Y25
            X40
            Y37
            X60
        G16 YZ ..... ;Depth profile
        G0 Y37 Z-20
        G1 Z-13
        G3 Y34 Z-10 J-3 K0


        G17 ..... ;C-type contour
        G0 X70 Y31 ..... ;Plane profile
        G1 Y40
            X80
            Y20
            X70
            Y31
        G16 XZ ..... ;Depth profile
        G0 X70 Z-20
N500  G1 X65 Z-10 ..... ;End of pocket geometry definition
```

Example 4:



To define the island 10 contours are used as shown here:

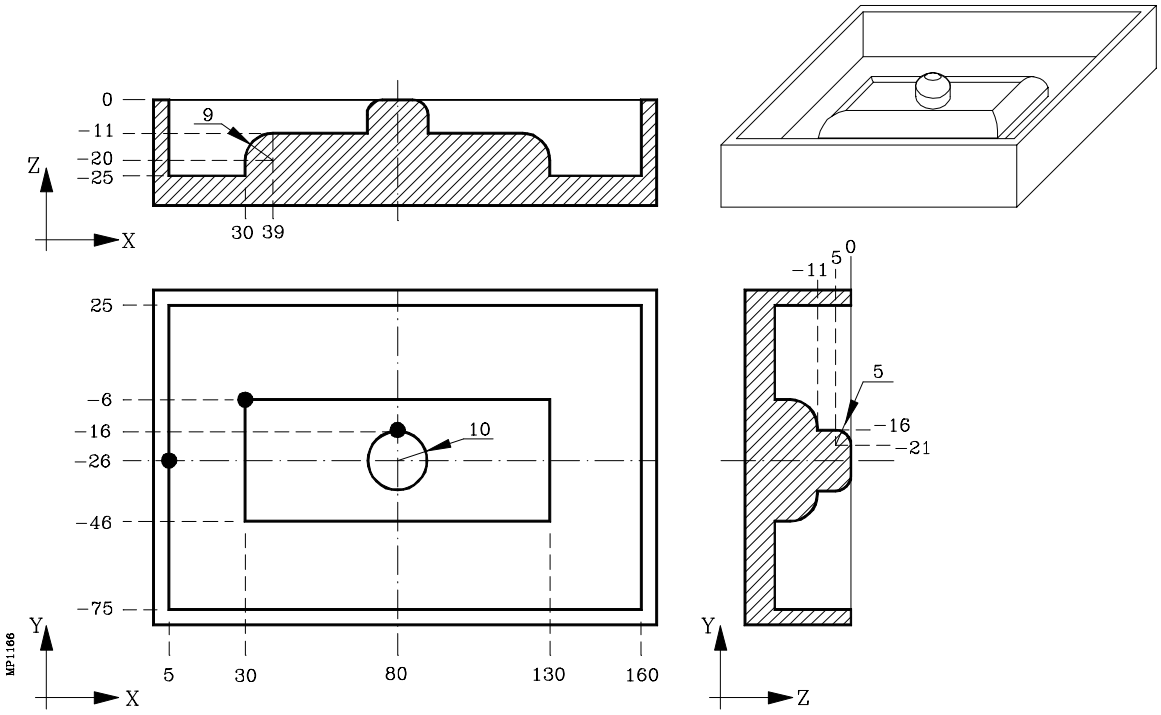




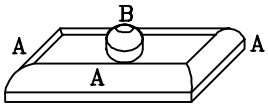

```
(TOR1=4,TOI1=0,TOR2=2.5,TOI2=0)
G17 G0 G43 G90 Z25 S1000 M3
G66 R200 C250 F300 S400 E500 ..... ;Definition of the 3D pocket
M30

N200 G67 B5 C0 I-30 R5 V100 F700 T1D1 M6 ..... ;Roughing  Operation
N250 G67 B1.15 I-29 R5 V100 F850 T1D1 M6 ..... ;Semi-finishing Operation
N300 G68 B1.5 L0.25 Q0 I-30 R5 V100 F500 T2D2 M6 ... ;Finishing  Operation

N400 G17 ..... ;Beginning of the pocket geometry definition
     G90 G0 X-70 Y20 Z0 ..... ;Outside contour (plane profile)
     G1  X70
         Y-90
         X-70
         Y20

     G17 ..... ;Contour number 1
     G90 G0 X42.5 Y5 ..... ;Plane profile
     G1 G91 X-16
         Y-60
         X32
         Y60
         X-16
     G16YZ ..... ;Depth profile
     G0 G90 Y5 Z-30
     G3 Y-25 Z0 J-30 K0

     G17 ..... ;Contour number 2
     G0 X27.5 Y-25
     G1 G91 Y31
     G1  X-2
         Y-62
         X2
         Y31
     G16XZ ..... ;Depth profile
     G0 G90 X27.5 Z-30
     G1 Z0
```

```
G17 ..... ;Contour number 3
G0 X57.5 Y-25
G1 G91 Y-31
     X2
     Y62
     X-2
     Y-31
G16XZ ..... ;Depth profile
G0 G90 X57.5 Z-30
G1 Z0

G17 ..... ;Contour number 4
G0 X0 Y-75
G1 G91 X-31
     Y-2
     X62
     Y2
     X-31
G16YZ ..... ;Depth profile
G0 G90 Y-75 Z-30
G1 Z0

G17 ..... ;Contour number 5
G0 X-30 Y-60
G1 G91 Y-16
     X60
     Y32
     X-60
     Y-16
G16XZ ..... ;Depth profile
G0 G90 X-30 Z-30
G2 X0 Z0 I30 K0

G17 ..... ;Contour number 6
G0 X0 Y-45
G1 G91 X31
     Y2
     X-62
     Y-2
     X31
G16YZ ..... ;Depth profile
G0 G90 Y-45 Z-30
G1 Z0

G17 ..... ;Contour number 7
G0 X-57.5 Y-25
G1 G91 Y31
     X-2
     Y-62
     X2
     Y31
G16XZ ..... ;Depth profile
G0 G90 X-57.5 Z-30
G1 Z0

G17 ..... ;Contour number 8
G0 X-42.5 Y5
G1 G91 X-16
     Y-60
     X32
     Y60
     X-16
G16YZ
G0 G90 Y5 Z-30
G3 Y-25 Z0 J-30 K0

G17 ..... ;Contour number 9
G0 X-27.5 Y-25
G1 G91 Y-31
     X2
     Y62
     X-2
     Y-31
G16XZ ..... ;Depth profile
G0 G90 X27.5 Z-30
G1 Z0

G17 ..... ;Contour number 10
G0 X0 Y0
G1   X-28
     Y-50
     X28
     Y0
     X0
G16YZ ..... ;Depth profile
     G0 Y0 Z-30
N500 G3 Y-25 Z-5 J-25 K0
```

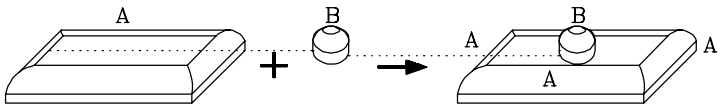
Example 5:



In this example, the island has 2 types of depth profiles: A and B.



2 contours are used to define the island: the low contour (A-type) and the high contour (B-type).




```
(TOR1=2.5,TOL1=20,TOI1=0,TOK1=0)
G17 G0 G43 G90 Z50 S1000 M4
G5
G66 R200 C250 F300 S400 E500 ..... ;3D pocket definition
M30

N200  G67 B5 C4 I-25 R5 V100 F400 T1D1 M6 ..... ;Roughing  operation
N250  G67 B2 I-25 R5 V100 F550 T2D1 M6 ..... ;Semi-finishing  operation
N300  G68 B1.5 L0.75 Q0 I-25 R5 V100 F275 T3D1 M6 ..... ;Finishing  operation
```

```
N400  G17 ..... ;Beginning of pocket geometry definition
      G90 G0 X5 Y-26 Z0 ..... ;Outside contour (plane profile)
      G1  Y25
          X160
          Y-75
          X5
          Y-26

      G17 ..... ;Low contour (A type)
      G90 G0 X30 Y-6 ..... ;Plane profile
      G1  Y-46
          X130
          Y-6
          X30
      G16 XZ ..... ;Depth profile
      G0  X30 Z-25
      G1  Z-20
      G2  X39 Z-11 I9 K0

      G17 ..... ;High contour (B-type)
      G90 G0 X80 Y-16 ..... ;Plane profile
      G2  I0 J-10
      G16 YZ ..... ;Depth profile
      G0  Y-16 Z-11
      G1  Y-16 Z-5
N500  G3  Y-21 Z0 J-5 K0 ..... ;End of pocket geometry definition
```

11.2.10 ERRORS

The CNC will issue the following errors:

ERROR 1025 : A tool of no radius has been programmed.
When using a tool with "0" radius while machining a pocket.

ERROR 1026 : A step greater than the tool diameter has been programmed.
When parameter "C" of the roughing operation is greater than the diameter of the roughing tool.

ERROR 1041 : A mandatory parameter not programmed in the canned cycle.
It comes up in the following instances:
- When parameters "I" and "R" have not been programmed in the roughing operation.
- When not using a roughing operation and not programming the "I" and "R" parameters for the semi-finishing operation.
- When not using a semi-finishing operation and not programming the "I" and "R" parameters for the finishing operation.
- When parameter "B" has not been programmed in the finishing operation.

ERROR 1042 : Wrong canned cycle parameter value.
It comes up in the following instances:
- When parameter "Q" of the finishing operation has the wrong value.
- When parameter "B" of the finishing operation has a "0" value.
- When parameter "J" of the finishing operation has been programmed with a value greater than the finishing tool radius.

ERROR 1043 : Wrong depth profile in an irregular pocket with islands
It comes up in the following instances:
- When the depth profiles of 2 sections of the same contour (simple or composite) cross each other
- When the finishing operation cannot be performed with the programmed tool. A typical case is a spherical mold with a non-spherical tool (parameter "J" not equal to the radius).

ERROR 1044 : The plane profile intersects itself in an irregular pocket with islands.
It comes up when any of the plane profiles of the programmed contours intersects itself.

ERROR 1046 : Wrong tool position prior to the canned cycle.
It comes up when calling the G66 cycle if the tool is positioned between the reference plane and the depth coordinate (bottom) of any of the operations.

ERROR 1047 : Open plane profile in an irregular pocket with islands.
It comes up when any of the programmed contours does not begin and end at the same point. It may be because G1 has not been programmed after the beginning, with G0, on any of the profiles.

ERROR 1048 : The part surface coordinate (top) has not been programmed in an irregular pocket with islands.
It comes up when the first point of the geometry does not include the pocket top coordinate.

ERROR 1049 : Wrong reference plane coordinate for the canned cycle.

It comes up when the coordinate of the reference plane is located between the part's "top" and "bottom" in any of the operations.

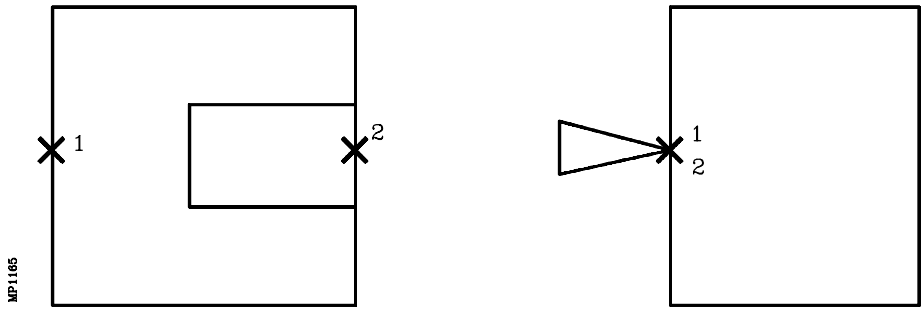
ERROR 1084 : Wrong circular path.

It comes up when any of the paths programmed in the geometry definition of the pocket is wrong.

ERROR 1227 : Wrong profile intersection in an irregular pocket with islands.

It comes up in the following instances:

- When two plane profiles have a common section (drawing on the left).
- When the initial points of two profiles in the main plane coincide (drawing on the right).



12. *WORKING WITH A PROBE*

The CNC has two probe inputs, one for TTL-type 5V DC signals and another for 24 V DC signals.

The connection of the different types of probes to these inputs are explained in the appendix to the Installation and Start-up manual.

This control allows the following operations to be performed, by using probes:

* Programming probing blocks with functions G75 and G76.

* Several tool calibration and part-measurement cycles by means of high-level language programming.

12.1 PROBING (G75,G76)

The **G75** function allows movements to be programmed which will end after the CNC **receives** the signal from the measuring probe used.

The **G76** function allows movements to be programmed which will end after the CNC **no longer receives** the signal from the measuring probe used.

Their definition format is:

G75 X..C # 5.5
G76 X..C # 5.5

After **G75** or **G76**, the required axis or axes will be programmed, as well as the coordinates of these axes which will define the end point of the programmed movement.

The machine will move according to the programmed path until it receives the signal from the probe (G75) or until it no longer receives the probe signal (G76). At this time, the CNC will consider the block finished, taking as the theoretical position of the axes the real position which they have at that time.

If the axes reach the programmed position before receiving (G75) or while receiving (G76) the external signal from the probe, the CNC will stop the movement of the axes.

This type of movement with probing blocks are very useful when it is required to generate measurement or verification programs for tools and parts.

Functions G75 and G76 are not modal and, therefore, must be programmed whenever it is wished to probe.

It is not possible to vary the Feedrate Override while either G75 or G76 is active. It stays set at 100 %.

Functions G75 and G76 are incompatible with each other and with G00, G02, G03, G33, G41 and G42 functions. In addition, once this has been performed, the CNC will assume functions G01 and G40.

12.2 PROBING CANNED CYCLES

The CNC has the following probing canned cycles:

1 Tool length calibration canned cycle.

2 Probe calibration canned cycle

3 Surface measuring canned cycle

4 Outside corner measuring canned cycle

5 Inside corner measuring canned cycle

6 Angle measuring canned cycle

7 Corner and angle measuring canned cycle

8 Hole measuring canned cycle

9 Boss measuring canned cycle

All the movements of these probing canned cycles will be performed in the X, Y, Z axes and the work plane must be formed by 2 of these axes (XY, XZ, YZ, YX, ZX, ZY). The other axis, which must be perpendicular to this plane, must be selected as the longitudinal axis.

Canned cycles will be programmed by means of the high level mnemonic, **PROBE**, which has the following programming format:

(PROBE(expression),(assignment statement),...)

This statement calls the probing cycle indicated by means of a number or any expression which results in a number. Besides, it allows the parameters of this cycle to be initialized with the values required to perform it, by means of assignment statements.

General considerations

Probing canned cycles are not modal, and therefore must be programmed whenever it is required to perform any of them.

The probes used in the performance of these cycles are:

* Probe placed on a fixed position on the machine, used for calibrating tools.

* Probe placed in the spindle, will be treated as a tool and will be used in the different measuring cycles.

The execution of a probing canned cycle does not alter the history of previous “G” functions, except for the radius compensation functions G41 and G42.

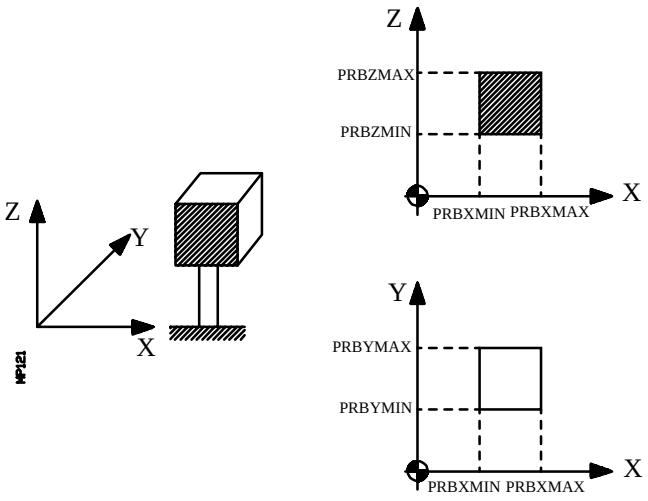
12.3 TOOL LENGTH CALIBRATION CANNED CYCLE

This is used to calibrate the length of the selected tool. Once the cycle has ended, the value (L) corresponding to the tool offset which is selected will be updated on the tool offset table.

To perform this cycle it is necessary to have a table-top probe, installed in a fixed position on the machine and with its faces parallel to axes X, Y, Z.

Its position will be indicated in absolute coordinates with respect to machine zero by means of the general machine parameters:

PRBXMIN	Indicates the minimum coordinate occupied by the probe along the X axis.
PRBXMAX	Indicates the maximum coordinate occupied by the probe along the X axis.
PRBYMIN	Indicates the minimum coordinate occupied by the probe along the Y axis.
PRBYMAX	Indicates the maximum coordinate occupied by the probe along the Y axis.
PRBZMIN	Indicates the minimum coordinate occupied by the probe along the Z axis.
PRBZMAX	Indicates the maximum coordinate occupied by the probe along the Z axis.



If it is the first time that the tool length has been calibrated, it is advisable to include an approximate value of its length (L) in the tool offset table.

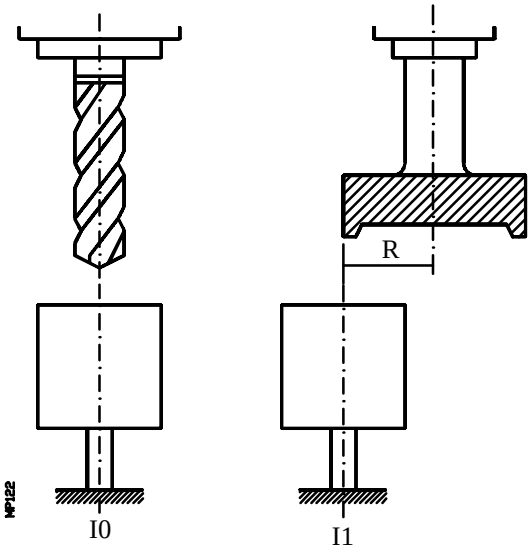
The programming format for this cycle is as follows:

(PROBE 1, B, I, F)

B5.5 Defines the safety distance. It must be programmed with a positive value and greater than 0.

I Indicates how the calibration canned cycle will be executed.

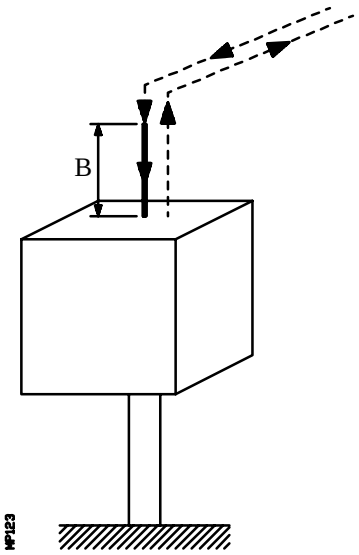
0= Tool calibration on its center.
1= Tool calibration on its end.



If this is not programmed, the cycle will take the IO value.

F5.5 Defines probing feedrate in mm/min or inch/min.

Basic operation:



1.- Approach

Movement of the probe in rapid (G00) from the point where the cycle is called to the approach point.

This point is to be found opposite the point where it is wished to measure, at a safety distance (B) from it and along the longitudinal axis.

The approaching movement is made in two stages:

1st Movement in the main work plane.
2nd Movement along the longitudinal axis.

2.- Probing

Movement of the probe along the longitudinal axis at the indicated feedrate (F), until the probe signal is received.

The maximum distance to be travelled in the probing movement is 2B. If, after travelling that distance, the CNC does not receive the probe signal, it will display the corresponding error code and stop the movement of the axes.

3.- Withdrawal

Movement of the probe in rapid (G00) from the point where it probed to the point where the cycle was called.

The withdrawal movement is made in two stages:

1st Movement along the longitudinal axis to the coordinate of the point (along this axis) from where the cycle was called.

2nd Movement in the main work plane to the point where the cycle is called.

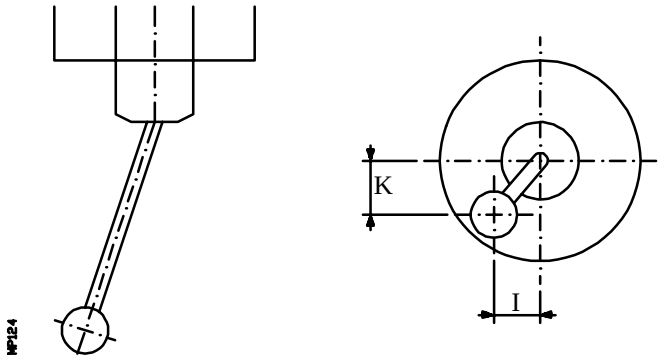
Once the cycle has been completed, the CNC will have updated the tool offset selected at the time on the tool offset table, value (L) and initialized the value of (K) to 0, it also returns the value of the global arithmetic parameter:

P299 Error detected. Difference between the measured tool length and the one assigned to it in the table.

12.4 PROBE CALIBRATING CANNED CYCLE

This is used to calibrate the probe situated in the spindle. This probe which previously must be calibrated in length, will be the one used in probe measuring canned cycles.

The cycle measures the deviation which the probe ball axis has with respect to the tool holder axis, using a previously machined hole with known center and dimensions for its calibration.



The CNC will treat each measuring probe used as just one more tool. The tool offset table fields corresponding to each probe will have the following meaning:

R Radius of the sphere (ball) of the probe. This value will be loaded into the table manually.

L Length of the probe. This value will be indicated by the tool length calibration cycle.

I Deviation of the probe ball with respect to the tool-holder axis, along the abscissa axis. This value will be indicated by the cycle.

K Deviation of the probe ball with respect to the tool holder axis, along the ordinate axis. This value will be indicated by the cycle.

The following steps will be followed for its calibration:

1.- Once the characteristics of the probe have been consulted, the value for the sphere radius (R) will be entered manually in the corresponding tool offset.

2.- After selecting the corresponding tool number and tool offset the Tool Length Calibration Cycle will be performed, the value of (L) will be updated and the value of (K) will be initialized to 0.

3.- Execution of the probe calibration canned cycle, updating the "I" and "K" values.

The programming format for this cycle is:

(PROBE 2,X,Y,Z,B,J,E,H,F)

X+/-5.5 Real coordinate, along the X axis, of the hole center.

Y+/-5.5 Real coordinate, along the Y axis, of the hole center.

Z+/-5.5 Real coordinate, along the Z axis, of the hole center.

B5.5 Defines the safety distance. Must be programmed with a positive value and over 0.

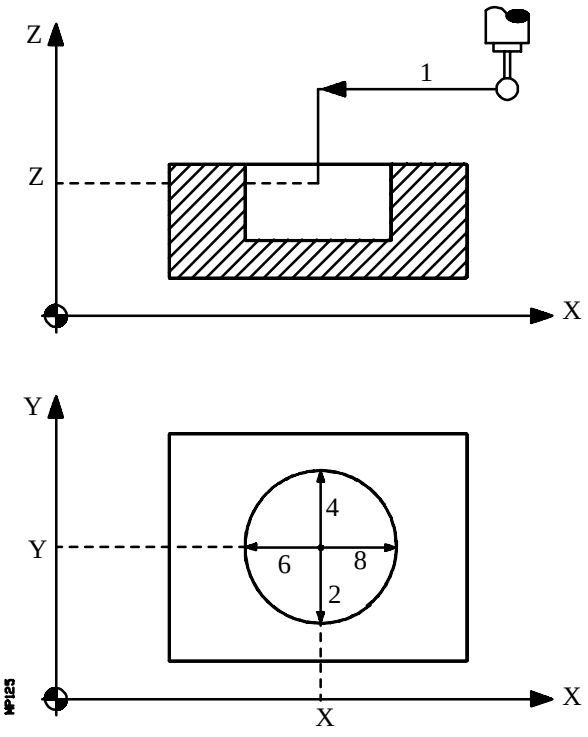
J5.5 Defines the real diameter of the hole. Must be programmed with a positive value and over 0.

E.5.5 Defines the distance which the probe moves back after initial probing. Must be programmed with a positive value and over 0.

H5.5 Defines the feedrate for the initial probing movement. Must be programmed in mm/minute or in inches/minute.

F5.5 Defines the probing feedrate. Must be programmed in mm/minute or in inches/minute.

Basic operation:



1.- Approach

Movement of the probe in rapid (G00) from the point where the cycle is called to the center of the hole.

The approaching movement is made in two stages:

1st Movement in the main work plane.

2nd Movement along the longitudinal axis.

2.- Probing

This movement consists of:

* Movement of the probe along the ordinate axis at the indicated feedrate (H), until the probe signal is received.

  The maximum distance to be travelled in the probing movement is "B+(J/2)". If, after travelling that distance, the CNC does not receive the probe signal, it will display the corresponding error code and stop the movement of the axes.

* Return of the probe in rapid (G00) the distance indicated in (E).

* Movement of the probe along the ordinate axis at the indicated feedrate (F), until the probe signal is received.

3.- Withdrawal

Movement of the probe in rapid (G00) from the point where it probed to the real center of the hole.

4.- Second probing movement.

Same as above.

5.- Withdrawal

Movement of the probe in rapid (G00) from the point where it probed to the real center of the hole along the ordinate axis.

6.- Third probing movement.

Same as above.

7.- Withdrawal

Movement of the probe in rapid (G00) from the point where it probed to the real center of the hole.

8.- Fourth probing movement.

Same as above.

9.- Withdrawal

This movement consists of:

* Movement of the probe in rapid (G00) from the point where it probed to the real center of the hole.

* Movement along the longitudinal axis to the coordinate of the point (along this axis) from where the cycle was called.

* Movement in the main work plane to the point where the cycle was called.

Once the cycle has been completed, the CNC will have updated the “I” and “K” values corresponding to the tool offset selected at the time on the tool offset table.

On the other hand, arithmetic parameter P299 returns the best value to be assigned to general machine parameter PRODEL.

12.5 SURFACE MEASURING CANNED CYCLE

A probe placed in the spindle will be used, which must be previously calibrated by means of canned cycles:

Canned cycle for calibrating tool length.
Canned cycle for calibrating probe.

This cycle allows correcting the value of the tool offset of the tool which has been used in the surface machining process. This correction will be used only when the measurement error exceeds a programmed value.

The programming format for this cycle is:

(PROBE 3,X,Y,Z,B,K,F,C,D,L)

X+/-5.5 Theoretical coordinate, along the X axis, of the point over which it is required to measure.

Y+/-5.5 Theoretical coordinate, along the Y axis, of the point over which it is required to measure.

Z+/-5.5 Theoretical coordinate, along the Z axis, of the point over which it is required to measure.

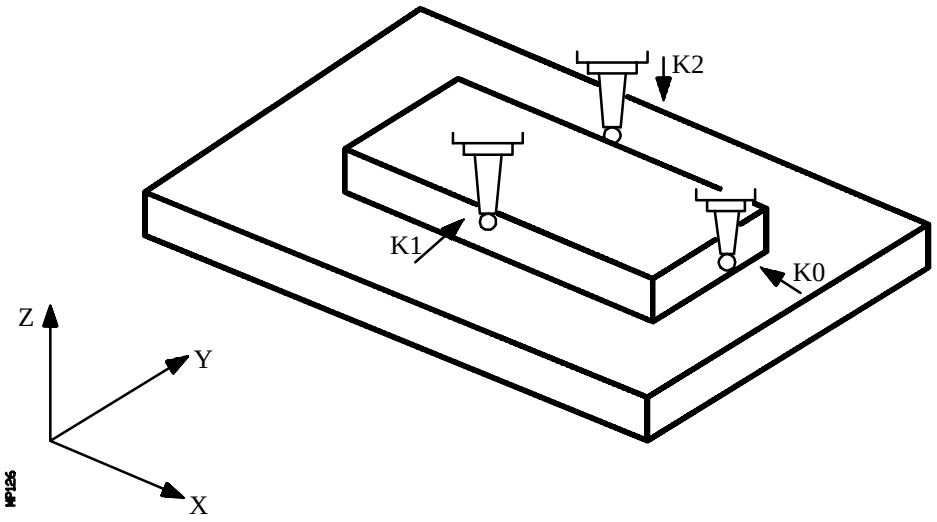
B5.5 Defines the safety distance. Must be programmed with a positive value and over 0.

The probe must be placed, with respect to the point to be measured, at a distance greater than this value when the cycle is called.

K Defines the axis with which it is required to measure the surface and will be defined by means of the following code:

0= With the abscissa axis of the work plane.
1= With the ordinate axis of the work plane.
2= With the longitudinal axis of the work plane.

If this is not programmed, the canned cycle will take the value of K0.



F5.5 Defines the probing feedrate in mm/min. or inches/min.

C Indicates where the probing cycle must finish.

0= Will return to the same point where the call to the cycle was made.
1= The cycle will finish over the measured point returning the longitudinal axis to the cycle calling point.

If this is not programmed, the canned cycle will take the value of C0.

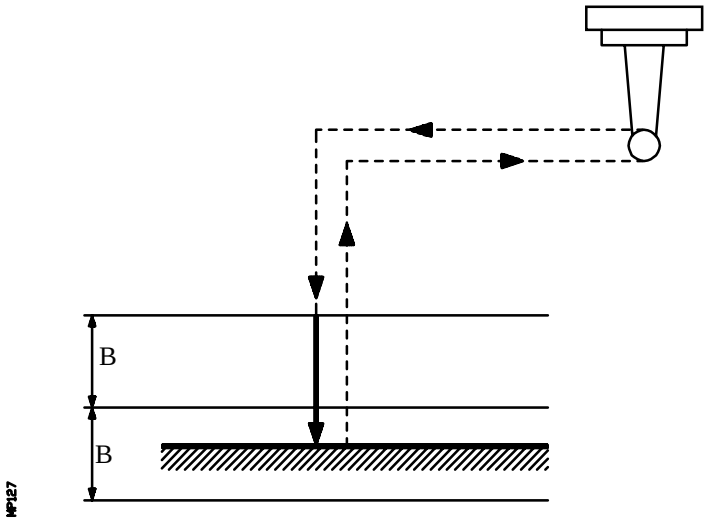
D4 Defines the number of the tool offset to be corrected, once the measurement cycle is completed.

If this is not programmed or is programmed with a value of 0, the CNC will understand that it is not required to make this correction.

L5.5 Defines the tolerance which will be applied to the error measured. It will be programmed with an absolute value and the tool offset will be corrected only when the error exceeds this value.

If this is not programmed, the canned cycle will take the value of 0.

Basic operation:



1.- Approach

Movement of the probe in rapid (G00) from the point where the cycle is called to the approach point.

This point is to be found opposite the point where it is wished to measure, at a safety distance (B) from this and along the probing axis (K).

The approaching movement is made in two stages:

1st Movement in the main work plane.
2nd Movement along the longitudinal axis.

2.- Probing

Movement of the probe along the selected axis (K) at the indicated feedrate (F), until the probe signal is received.

The maximum distance to be travelled in the probing movement is 2B. If, after travelling that distance, the CNC does not receive the probe signal, it will display the corresponding error code and stop the movement of the axes.

Once probing has been made, the CNC will assume as their theoretical position the real position of the axes when the probe signal is received .

3. Withdrawal

Movement of the probe in rapid (G00) from the point where it probed to the point where the cycle was called.

The withdrawal movement is made in three stages:

1st Movement along the probing axis to the approach point.

2nd Movement along the longitudinal axis to the coordinate of the point (along this axis) from where the cycle was called.

3rd When (C0) is programmed, movement is made in the main work plane to the point where the cycle is called.

Once the cycle has been completed, the CNC will return the real values obtained after measurement, in the following global arithmetic parameters.

P298 **Real surface coordinate.**

P299 **Error detected.** Difference between the real coordinate of the surface and the theoretical programmed coordinate.

If the Tool Offset Number (D) was selected, the CNC will modify the values of this tool offset, whenever the measurement error is equal to or greater than the tolerance (L).

Depending on the axis the measurement is made with (K), the correction will be made on the length or radius value.

* If the measurement is made with the axis longitudinal to the work plane, the length wear (K) of the indicated tool offset (D) will be modified.

* If the measurement is made with one of the axes which make up the work plane, the radius wear (I) of the indicated tool offset (D) will be modified.

12.6 OUTSIDE CORNER MEASURING CANNED CYCLE

A probe placed in the spindle will be used, which must be previously calibrated by means of canned cycles:

Canned cycle for calibrating tool length.
Canned cycle for calibrating probe.

The programming format for this cycle is:

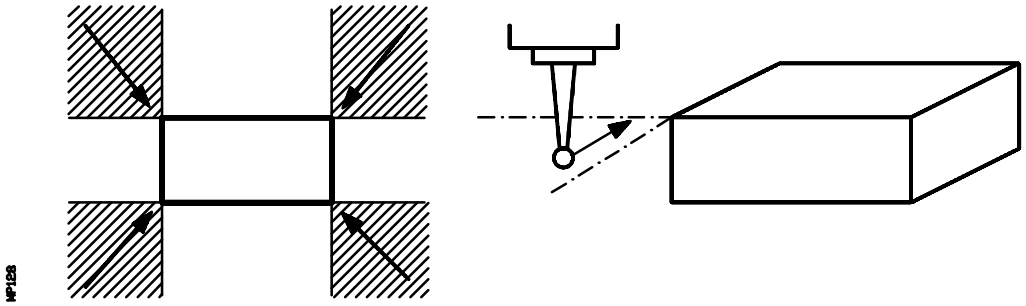
(PROBE 4,X,Y,Z,B,F)

X+/-5.5 Theoretical coordinate, along the X axis, of the corner to be measured.

Y+/-5.5 Theoretical coordinate, along the Y axis, of the corner to be measured.

Z+/-5.5 Theoretical coordinate, along the Z axis, of the corner to be measured.

Depending on the corner of the part it is required to measure, the probe must be placed in the corresponding shaded area (see figure) before calling the cycle.

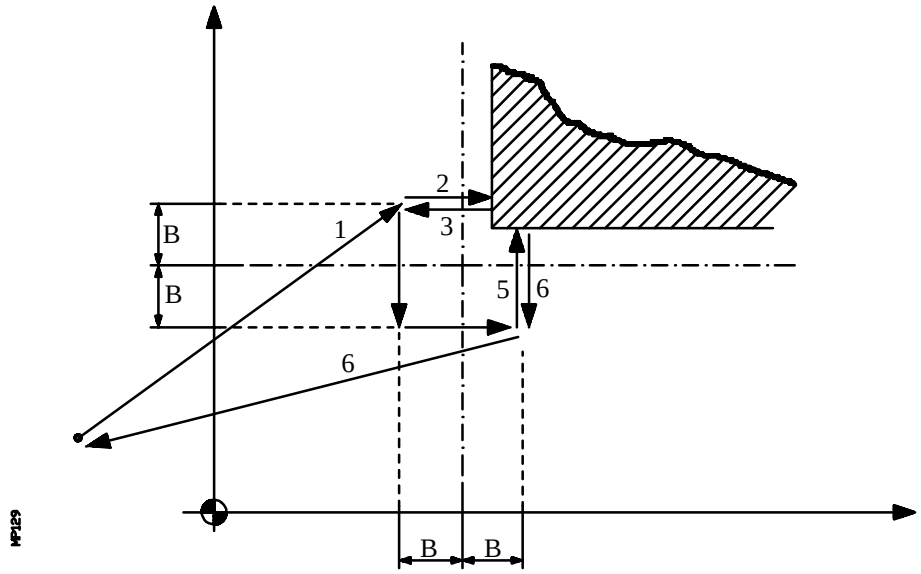


B5.5 Defines the safety distance. Must be programmed with a positive value and over 0.

The probe must be placed, with respect to the point to be measured, at a distance greater than this value when the cycle is called.

F5.5 Defines the probing feedrate in mm/min or inch/min.

Basic operation:



1.- Approach

Movement of the probe in rapid (G00) from the point where the cycle is called to the first approach point, situated at a distance (B) from the first face to be probed.

The approaching movement is made in two stages:

1st Movement in the main work plane.
2nd Movement along the longitudinal axis.

2.- Probing

Movement of the probe along the abscissa axis at the indicated feedrate (F), until the probe signal is received.

The maximum distance to be travelled in the probing movement is 2B. If, after travelling that distance, the CNC does not receive the probe signal, it will display the corresponding error code and stop the movement of the axes.

3.- Withdrawal

Movement of the probe in rapid (G00) from the point where it probed to the first approach point

4.- Second approach

Movement of the probe in rapid (G00) from the first approach point to the second.

The approaching movement is made in two stages:

1st Movement along the ordinate plane.
2nd Movement along the abscissa axis.

5.- Second probing

Movement of the probe along the abscissa axis at the indicated feedrate (F), until the probe signal is received.

The maximum distance to be travelled in the probing movement is 2B. If, after travelling that distance, the CNC does not receive the probe signal, it will display the corresponding error code and stop the movement of the axes.

6.- Withdrawal

Movement of the probe in rapid (G00) from the point where it probed for the second time to the point where the cycle was called.

The withdrawal movement is made in three stages:

1st Movement along the probing axis to the second approach point.

2nd Movement along the longitudinal axis to the coordinate of the point corresponding to this axis where the cycle is called.

3rd Movement in the main work plane to the point where the cycle is called.

Once the cycle has been completed, the CNC will return the real values obtained after measurement, in the following global arithmetic parameters.

P296 **Real coordinate of the corner along the abscissa axis.**

P297 **Real coordinate of the corner along the ordinate axis.**

P298 **Error detected along the abscissa axis.** Difference between the real coordinate of the corner and the theoretical programmed coordinate.

P299 **Error detected along the ordinate axis.** Difference between the real coordinate of the corner and the theoretical programmed coordinate.

12.7 INSIDE CORNER MEASURING CANNED CYCLE

A probe placed in the spindle will be used, which must be previously calibrated by means of canned cycles:

Canned cycle for calibrating tool length.
Canned cycle for calibrating probe.

The programming format for this cycle is:

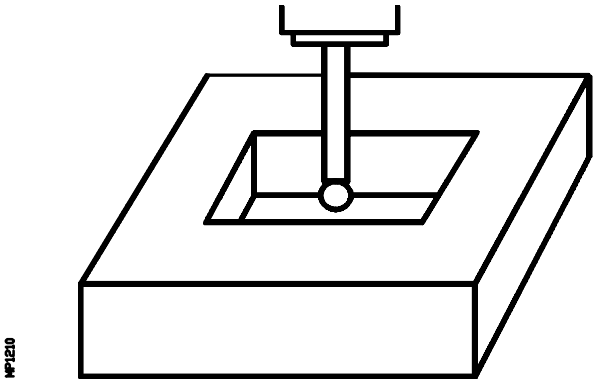
(PROBE 5,X,Y,Z,B,F)

X+/-5.5 Theoretical coordinate, along the X axis, of the corner to be measured.

Y+/-5.5 Theoretical coordinate, along the Y axis, of the corner to be measured.

Z+/-5.5 Theoretical coordinate, along the Z axis, of the corner to be measured.

The probe must be placed within the pocket before calling the cycle.

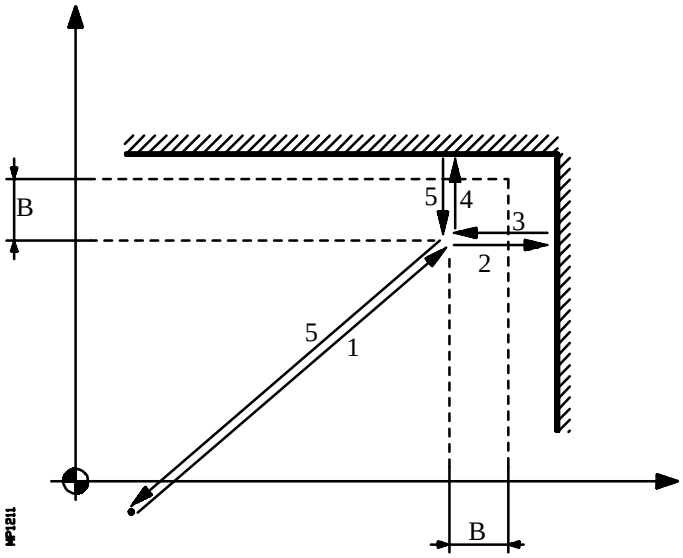


B5.5 Defines the safety distance. Must be programmed with a positive value and over 0.

The probe must be placed, with respect to the point to be measured, at a distance greater than this value when the cycle is called.

F5.5 Defines the probing feedrate in mm/min. or inch/min.

Basic operation:



1.- Approach

Movement of the probe in rapid (G00) from the point where the cycle is called to the first approach point, situated at a distance (B) from both faces to be probed.

The approaching movement is made in two stages:

1st Movement in the main work plane.
2nd Movement along the longitudinal axis.

2.- Probing

Movement of the probe along the abscissa axis at the indicated feedrate (F), until the probe signal is received.

The maximum distance to be travelled in the probing movement is 2B. If, after travelling that distance, the CNC does not receive the probe signal, it will display the corresponding error code and stop the movement of the axes.

3.- Withdrawal

Movement of the probe in rapid (G00) from the point where it probed to the approach point

4.- Second probing

Movement of the probe along the abscissa axis at the indicated feedrate (F), until the probe signal is received.

The maximum distance to be travelled in the probing movement is 2B. If, after travelling that distance, the CNC does not receive the probe signal, it will display the corresponding error code and stop the movement of the axes.

5.- Withdrawal

Movement of the probe in rapid (G00) from the point where it probed for the second time to the point where the cycle was called.

The withdrawal movement is made in three stages:

1st Movement along the probing axis to the approach point.

2nd Movement along the longitudinal axis to the coordinate of the point corresponding to this axis where the cycle is called.

3rd Movement in the main work plane to the point where the cycle is called.

Once the cycle has been completed, the CNC will return the real values obtained after measurement, in the following global arithmetic parameters.

P296 **Real coordinate of the corner along the abscissa axis.**

P297 **Real coordinate of the corner along the ordinate axis.**

P298 **Error detected along the abscissa axis.** Difference between the real coordinate of the corner and the theoretical programmed coordinate.

P299 **Error detected along the ordinate axis.** Difference between the real coordinate of the corner and the theoretical programmed coordinate.

12.8 ANGLE MEASURING CANNED CYCLE

A probe placed in the spindle will be used, which must be previously calibrated by means of canned cycles:

Canned cycle for calibrating tool length.
Canned cycle for calibrating probe.

The programming format for this cycle is:

(PROBE 6,X,Y,Z,B,F)

X+/-5.5 Theoretical coordinate, along the X axis, of the angle to be measured.

Y+/-5.5 Theoretical coordinate, along the Y axis, of the angle to be measured.

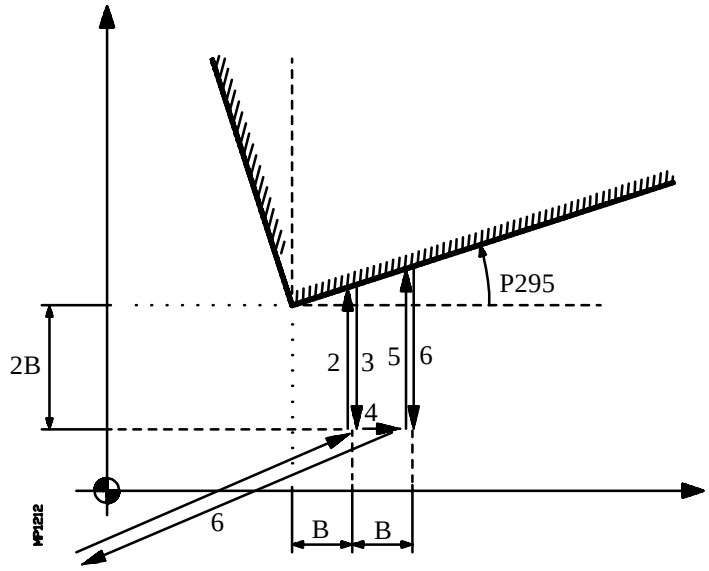
Z+/-5.5 Theoretical coordinate, along the Z axis, of the angle to be measured.

B5.5 Defines the safety distance. Must be programmed with a positive value and over 0.

The probe must be placed, with respect to the point to be measured, at a distance greater than double this value when the cycle is called.

F5.5 Defines the probing feedrate in mm/min. or inch/min.

Basic operation:



1.- Approach

Movement of the probe in rapid (G00) from the point where the cycle is called to the first approach point, situated at a distance (B) from the programmed vertex and at (2B) from the face to be probed.

The approaching movement is made in two stages:

1st Movement in the main work plane.
2nd Movement along the longitudinal axis.

2.- Probing

Movement of the probe along the ordinate axis at the indicated feedrate (F), until the probe signal is received.

The maximum distance to be travelled in the probing movement is 3B. If, after travelling that distance, the CNC does not receive the probe signal, it will display the corresponding error code and stop the movement of the axes.

3.- Withdrawal

Movement of the probe in rapid (G00) from the point where it probed to the first approach point

4.- Second approach

Movement of the probe in rapid (G00) from the first approach point to the second. It is at a distance (B) from the first one.

5.- Second probing

Movement of the probe along the abscissa axis at the indicated feedrate (F), until the probe signal is received.

The maximum distance to be travelled in the probing movement is 4B. If, after travelling that distance, the CNC does not receive the probe signal, it will display the corresponding error code and stop the movement of the axes.

6.- Withdrawal

Movement of the probe in rapid (G00) from the point where it probed for the second time to the point where the cycle was called.

The withdrawal movement is made in three stages:

1st Movement along the probing axis to the second approach point.

2nd Movement along the longitudinal axis to the coordinate of the point (along this axis) from where the cycle was called.

3rd Movement in the main work plane to the point where the cycle is called.

Once the cycle has been completed, the CNC will return the real values obtained after measurement, in the following global arithmetic parameter.

P295 **Inclination angle** which the part has in relation to the abscissa axis.

This cycle allows angles between ±45° to be measured.

If the angle to be measured is ≥ 45°, the CNC will display the corresponding error.

If the angle to be measured is ≤-45°, the probe will collide with the part.

12.9 OUTSIDE CORNER AND ANGLE MEASURING CANNED CYCLE

A probe placed in the spindle will be used, which must be previously calibrated by means of canned cycles:

Canned cycle for calibrating tool length.
Canned cycle for calibrating probe.

The programming format for this cycle is:

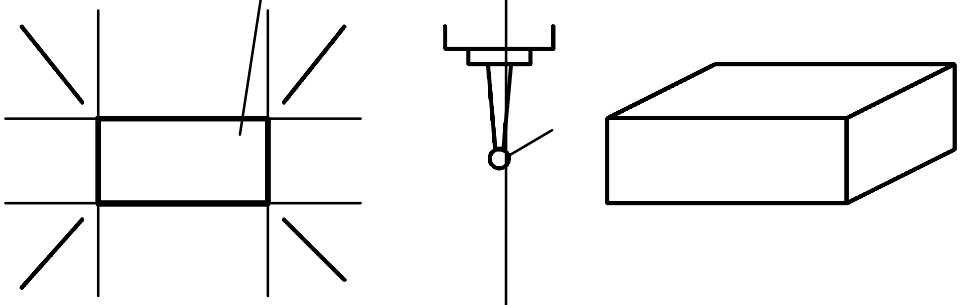
(PROBE 7,X,Y,Z,B,F)

X±5.5 Theoretical coordinate, along the X axis, of the corner to be measured.

Y±5.5 Theoretical coordinate, along the Y axis, of the corner to be measured.

Z±5.5 Theoretical coordinate, along the Z axis, of the corner to be measured.

Depending on the corner of the part it is required to measure, the probe must be placed in the corresponding shaded area (see figure) before calling the cycle.

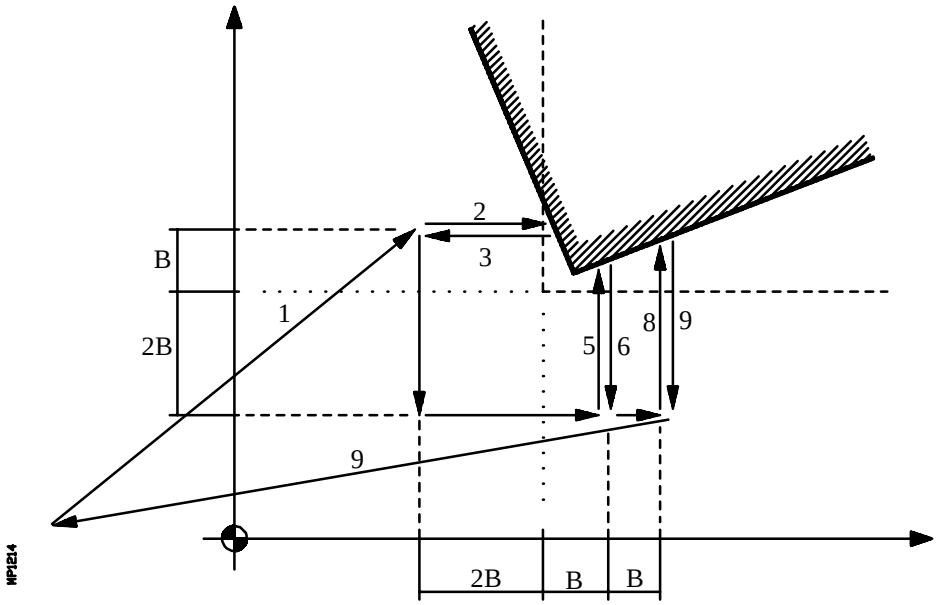


B5.5 Defines the safety distance. Must be programmed with a positive value and over 0.

The probe must be placed, with respect to the point to be measured, at a distance greater than double this value when the cycle is called.

F5.5 Defines the probing feedrate in mm/min. ~~or inch/min.~~

Basic operation:



1.- Approach

Movement of the probe in rapid (G00) from the point where the cycle is called to the first approach point, situated at a distance (B) from the first face to be probed.

The approaching movement is made in two stages:

1st Movement in the main work plane.
2nd Movement along the longitudinal axis.

2.- Probing

Movement of the probe along the abscissa axis at the indicated feedrate (F), until the probe signal is received.

The maximum distance to be travelled in the probing movement is 3B. If, after travelling that distance, the CNC does not receive the probe signal, it will display the corresponding error code and stop the movement of the axes.

3.- Withdrawal

Movement of the probe in rapid (G00) from the point where it probed to the first approach point.

4.- Second approach

Movement of the probe in rapid (G00) from the first approach point to the second, situated at a distance (2B) from the second face to be probed.

The approaching movement is made in two stages:

1st Movement along the ordinate plane.
2nd Movement along the abscissa axis.

5.- Second probing

Movement of the probe along the abscissa axis at the indicated feedrate (F), until the probe signal is received.

The maximum distance to be travelled in the probing movement is 3B. If, after travelling that distance, the CNC does not receive the probe signal, it will display the corresponding error code and stop the movement of the axes.

6.- Withdrawal

Movement of the probe in rapid (G00) from the point where it probed to the second point of approach.

7.- Third approach

Movement of the probe in rapid (G00) from the second approach point to the third, situated at a distance (B) from the previous point.

8.- Third probing

Movement of the probe along the ordinate axis at the indicated feedrate (F), until the probe signal is received.

The maximum distance to be travelled in the probing movement is 4B. If, after travelling that distance, the CNC does not receive the probe signal, it will display the corresponding error code and stop the movement of the axes.

9.- Withdrawal

Movement of the probe in rapid (G00) from the third probing point to the point where the cycle was called.

The withdrawal movement is made in three stages:

1st Movement along probing axis to the third approach point.

2nd Movement along the longitudinal axis to the coordinate of the point (along this axis) from where the cycle was called.

3rd Movement in the main work plane to the point where the cycle is called.

Once the cycle has been completed, the CNC will return the real values obtained after measurement, in the following global arithmetic parameter.

P295 **Inclination angle which the part has in relation to the abscissa axis.**

P296 **Real coordinate of the corner along the abscissa axis.**

P297 **Real coordinate of the corner along the ordinate axis.**

P298 **Error detected along the abscissa axis.** Difference between the real coordinate of the corner and the programmed theoretical coordinate.

P299 **Error detected along the ordinate axis.** Difference between the real coordinate of the corner and the programmed theoretical coordinate.

This cycle allows angles between ±45° to be measured.

If the angle to be measured is ≥ 45° the CNC will display the corresponding error.

If the angle to be measured is ≤-45°, the probe will collide with the part.

12.10 HOLE MEASURING CANNED CYCLE

A probe placed in the spindle will be used, which must be previously calibrated by means of canned cycles:

Canned cycle for calibrating tool length.
Canned cycle for calibrating probe.

The programming format for this cycle is:

(PROBE 8,X,Y,Z,B,J,E,C,H,F)

X±5.5 Theoretical coordinate, along the X axis, of the center of the hole.

Y±5.5 Theoretical coordinate, along the Y axis, of the center of the hole.

Z±5.5 Theoretical coordinate, along the Z axis, of the center of the hole.

B5.5 Defines the safety distance. Must be programmed with a positive value and over 0.

J5.5 Defines the theoretical diameter of the hole. Must be programmed with a positive value and over 0.

This cycle allows holes to be measured with diameters of no more than (J+B).

E.5.5 Defines the distance which the probe moves back after initial probing. Must be programmed with a positive value and over 0.

C Indicates where the probing cycle must finish.

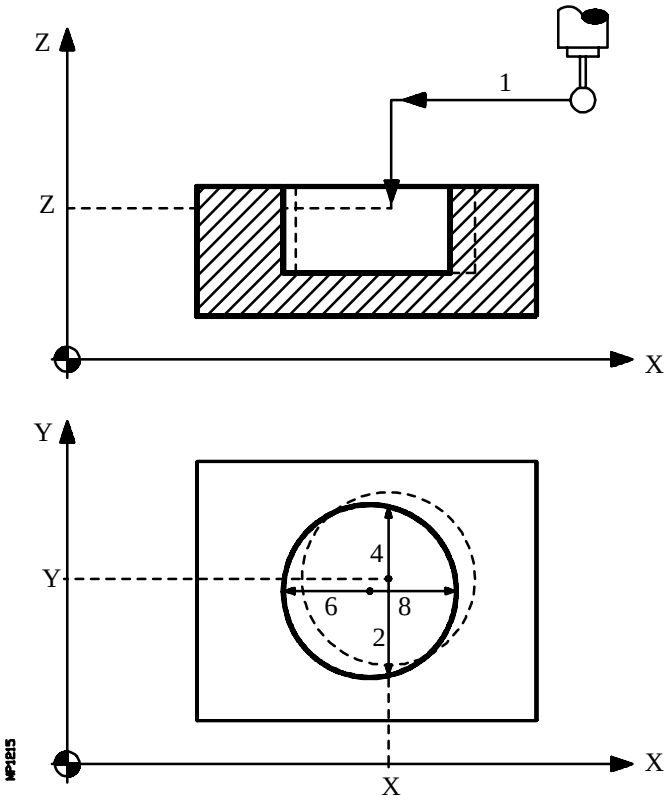
0= Will return to the same point where the call to the cycle was made.
1= The cycle will finish over the measured point returning the longitudinal axis to the cycle calling point.

If this is not programmed, the canned cycle will take the value of C0.

H5.5 Defines the initial probing feedrate in mm/min or in inch/min.

F5.5 Defines the probing feedrate in mm/min or inch/min.

Basic operation:



1.- Approach

Movement of the probe in rapid (G00) from the point where the cycle is called to the center of the hole.

The approaching movement is made in two stages:

1st Movement in the main work plane.
2nd Movement along the longitudinal axis.

2.- Probing

This movement consists of:

* Movement of the probe along the ordinate axis at the indicated feedrate (H), until the probe signal is received.

  The maximum distance to be travelled in the probing movement is "B+(J/2)". If, after travelling that distance, the CNC does not receive the probe signal, it will display the corresponding error code and stop the movement of the axes.

* Return of the probe in rapid (G00) the distance indicated in (E).

* Movement of the probe along the ordinate axis at the indicated feedrate (F), until the probe signal is received.

3.- Withdrawal

Movement of the probe in rapid (G00) from the point where it probed to the theoretical center of the hole.

4.- Second probing movement.

Same as above.

5.- Withdrawal

Movement of the probe in rapid (G00) from the point where it probed to the real center (calculated) of the hole along the ordinate axis.

6.- Third probing movement.

Same as above.

7.- Withdrawal

Movement of the probe in rapid (G00) from the point where it probed to the theoretical center of the hole.

8.- Fourth probing movement.

Same as above.

9.- Withdrawal

This movement consists of:

* Movement of the probe in rapid (G00) from the point where it probed to the real center (calculated) of the hole.

* Should (C0) be programmed, the probe will be moved to the point where the cycle was called.

  1st Movement along the longitudinal axis to the coordinate of the point (along this axis) from where the cycle was called.

  2nd Movement on the main work plane to the point where the cycle was called.

Once the cycle has been completed, the CNC will return the real values obtained after measurement, in the following global arithmetic parameter.

P294 **Hole diameter.**

P295 **Hole diameter error.** Difference between the real diameter and programmed diameter.

P296 **Real coordinate of the center along the abscissa axis.**

P297 **Real coordinate of the center along the ordinate axis.**

P298 **Error detected along the abscissa axis.** Difference between the real coordinate of the center and the programmed theoretical coordinate.

P299 **Error detected along the ordinate axis.** Difference between the real coordinate of the center and the programmed theoretical coordinate.

12.11 BOSS MEASURING CANNED CYCLE

A probe placed in the spindle will be used, which must be previously calibrated by means of canned cycles:

Canned cycle for calibrating tool length.
Canned cycle for calibrating probe.

The programming format for this cycle is:

(PROBE 9,X,Y,Z,B,J,E,C,H,F)

X±5.5 Theoretical coordinate, along the X axis, of the center of the boss.

Y±5.5 Theoretical coordinate, along the Y axis, of the center of the boss.

Z±5.5 Theoretical coordinate, along the Z axis, of the center of the boss.

B5.5 Defines the safety distance. Must be programmed with a positive value and over 0.

J5.5 Defines the theoretical diameter of the hole. Must be programmed with a positive value and over 0.

This cycle allows holes to be measured with diameters of no more than (J+B).

E.5.5 Defines the distance which the probe moves back after initial probing. Must be programmed with a positive value and over 0.

C Indicates where the probing cycle must finish.

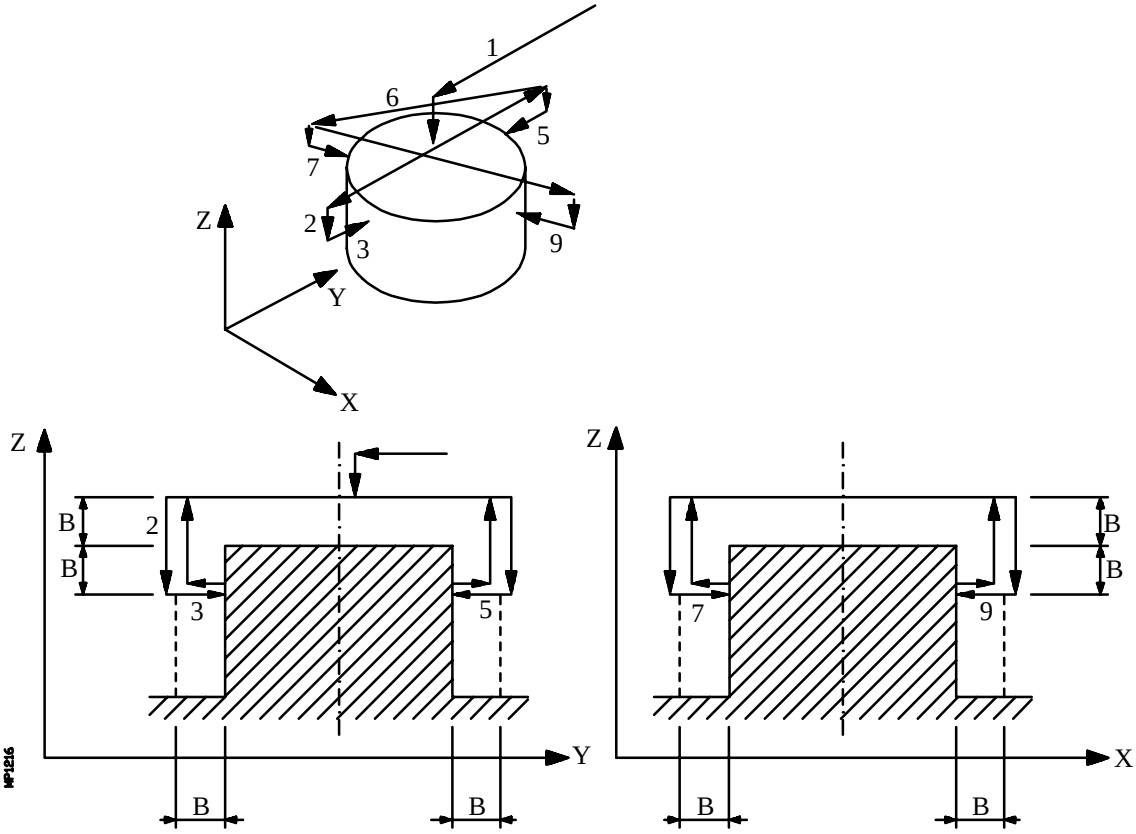
0= Will return to the same point where the call to the cycle was made.
1= The cycle will finish by positioning the probe over the center of the boss, at a distance (B) from the programmed theoretical coordinate.

If this is not programmed, the canned cycle will take the value of C0.

H5.5 Defines the feedrate for the initial probing movement. Must be programmed in mm/minute or in inches/minute.

F5.5 Defines probing feedrate in mm/min. or inch/min.

Basic operation:



1.- Positioning over the center of the boss.

Movement of the probe in rapid (G00) from the point where the cycle is called to the center of the boss.

The approaching movement is made in two stages:

1st Movement in the main work plane.
2nd Movement along the longitudinal axis up to a distance (B) from the programmed surface.

2.- Movement to the first approach point

This movement of the probe which is made in rapid (G00) consists of:

1st Movement along the ordinate axis.
2nd Movement of the longitudinal axis the distance (2B).

3.- Probing

This movement consists of:

* Movement of the probe along the ordinate axis at the indicated feedrate (H), until the probe signal is received.

  The maximum distance to be travelled in the probing movement is "B+(J/2)". If, after travelling that distance, the CNC does not receive the probe signal, it will display the corresponding error code and stop the movement of the axes.

* Return of the probe in rapid (G00) the distance indicated in (E).

* Movement of the probe along the ordinate axis at the indicated feedrate (F), until the probe signal is received.

4.- Movement to second approach point

This movement of the probe which is made in rapid (G00) consists of:

* Withdrawal to the first approach point.

* Movement to a distance (B) above the boss, to the second approach point.

5.- Second probing movement.

Same as the first probing.

6.- Third approach movement.

Same as above.

7.- Third probing movement.

Same as above.

8.- Fourth approach movement.

Same as above.

9.- Fourth probing movement.

Same as above.

10.- Withdrawal

This movement consists of:

* Withdrawal to the fourth approach point.

* Movement of the probe in rapid (G00) and at a distance (B) above the boss to the real center (calculated) of the boss.

* Should (C0) be programmed, the probe will be moved to the point where the cycle was called.

  1st Movement along the longitudinal axis to the coordinate of the point (along this axis) from where the cycle was called.

  2nd Movement on the main work plane to the point where the cycle was called.

Once the cycle has been completed, the CNC will return the real values obtained after measurement, in the following global arithmetic parameter.

P294 **Boss diameter.**

P295 **Boss diameter error.** Difference between the real diameter and programmed diameter.

P296 **Real coordinate of the center along the abscissa axis.**

P297 **Real coordinate of the center along the ordinate axis.**

P298 **Error detected along the abscissa axis.** Difference between the real coordinate of the center and the programmed theoretical coordinate.

P299 **Error detected along the ordinate axis.** Difference between the real coordinate of the center and the programmed theoretical coordinate.

13. PROGRAMMING IN HIGH-LEVEL LANGUAGE

The CNC has a series of internal variables which can be accessed from the user program, from the PLC program or through DNC.

Access to these variables from the user program is gained with high-level commands.

Each of the system variables which can be accessed will be referred to by means of its mnemonic, and will be separated, according to their use, into read-only variables and read-write variables.

13.1 LEXICAL DESCRIPTION

All the words which form the high-level language of the numerical control must be written in capital letters except for associated texts which may be written in upper and lower case letters.

The following elements are available for high-level programming:

- Reserved words.
- Numerical constants
- Symbols

13.1.1 RESERVED WORDS

The set of words which the CNC uses in high-level programming for naming system variables, operators, control mnemonics, etc. are as follows:

ANAIn	**ANAOn**	**BLKN**	**CALL**	**CALLP**	**CLOCK**
CNCERR	**CNCFRO**	**CNCSSO**	**CYTIME**	**DATE**	**DEFLEX**
DEFLEY	**DEFLEZ**	**DFHOLD**	**DIGIT**	**DIST(X-C)**	**DNCERR**
DNCF	**DNCFPR**	**DNCFRO**	**DNCS**	**DNCSL**	**DNCSSO**
DPOS(X-C)	**DSBLK**	**DSTOP**	**DW**	**EFHOLD**	**ERROR**
ESBLK	**ESTOP**	**EXEC**	**FEED**	**FIRST**	**FLWE(X-C)**
FLWES	**FOZLO(X-C)**	**FOZONE**	**FOZUP(X-C)**	**FPREV**	**FRO**
FZLO(X-C)	**FZONE**	**FZUP(X-C)**	**GGSA**	**GGSB**	**GGSC**
GGSD	**GMS**	**GOTO**	**GSn**	**GTRATY**	**GUP n**
IB	**IF**	**INPUT**	**KEY**	**KEYSRC**	**LONGAX**
LUP (a,b)	**MCALL**	**MDOFF**	**MIRROR**	**MP(X-C)n**	**MPASn**
MPGn	**MPLCn**	**MPSn**	**MPSSn**	**MSG**	**MSn**
NBTOOL	**NXTOD**	**NXTOOL**	**ODW**	**OPEN**	**OPMODA**
OPMODB	**OPMODC**	**OPMODE**	**ORG(X-C)**	**ORG(X-C)n**	**ORGROA**
ORGROB	**ORGROC**	**ORGROI**	**ORGROJ**	**ORGROK**	**ORGROQ**
ORGROR	**ORGROS**	**ORGROT**	**ORGROX**	**ORGROY**	**ORGROZ**
PAGE	**PARTC**	**PCALL**	**PLANE**	**PLCCn**	**PLCERR**
PLCF	**PLCFPR**	**PLCFRO**	**PLCIn**	**PLCMn**	**PLCMSG**
PLCOF(X-C)	**PLCOn**	**PLCRn**	**PLCS**	**PLCSL**	**PLCSSO**
PLCTn	**PORGF**	**PORGS**	**POS(X-C)**	**POSS**	**PPOS(X-C)**
PRBST	**PRGF**	**PRGFIN**	**PRGFPR**	**PRGFRO**	**PRGN**
PRGS	**PRGSL**	**PRGSSO**	**PROBE**	**REPOS**	**RET**
ROTPF	**ROTPS**	**RPOSS**	**RPT**	**RTPOSS**	**SCALE**
SCALE(X-C)	**SCNCSO**	**SDNCS**	**SDNCSL**	**SDNCSO**	**SFLWES**
SK	**SLIMIT**	**SPEED**	**SPLCS**	**SPLCSL**	**SPLCSO**
SPOSS	**SPRGS**	**SPRGSL**	**SPRGSO**	**SREAL**	**SRPOSS**
SRTPOS	**SSLIMI**	**SSO**	**SSPEED**	**SSREAL**	**SSSO**
STPOSS	**SUB**	**SYMBOL**	**SYSTEM**	**SZLO(X-C)**	**SZONE**
SZUP(X-C)	**TIME**	**TIMER**	**TLFDn**	**TLFFn**	**TLFNn**
TLFRn	**TMZPn**	**TMZTn**	**TOD**	**TOIn**	**TOKn**
TOLn	**TOOL**	**TOOROF**	**TOOROS**	**TORn**	**TPOS(X-C)**
TPOSS	**TRACE**	**TZLO(X-C)**	**TZONE**	**TZUP(X-C)**	**WBUF**
WBUF	**WKEY**	**WRITE**			

Words ending in (X-C) indicate a set of 9 elements formed by the corresponding root followed by X,Y,Z,U,V,W,A,B and C.

ORG(X-C)->ORGX,ORGY,ORGZ,ORGU,ORGV,ORGW,ORGA,ORGB,ORGC

All the letters of the alphabet A-Z are also reserved words, as they can make up a high-level language word when used alone.

13.1.2 NUMERICAL CONSTANTS

The blocks programmed in high-level language allow numbers in decimal format which do not exceed the format ±6.5 and numbers in hexadecimal format, in which case they must be preceded by the **$** sign, with a maximum of 8 digits.

The assignment to a variable of a constant higher than the format ±6,5 will bemade by means of arithmetic parameters, by means of arithmetic expressions or by means of constants expressed in hexadecimal format.

Example: To assign the value 100000000 to the variable "TIMER", It can be done in one of the following ways:

```
(TIMER  = $5F5E100)
(TIMER  = 10000 * 10000)
(P100   = 10000 * 10000)
(TIMER  = P100)
```

When the CNC is working in metric system (mm) resolution is in tenths of a micron, and figures are programmed in the format ±5.4 (positive or negative, with 5 integers and 4 decimals), and if the CNC is operating in inches, resolution is in 0.00001 inches, figures being programmed with the format ±4.5 (positive or negative, with 4 integers and 5 decimals).

For the convenience of the programmer, this control always allows the format ±5.5 (positive or negative, with 5 integers and 5 decimals), adjusting each number appropriately to the working units every time they are used.

13.1.3. SYMBOLS

The symbols used in high-level language are:

( ) " = + - * / ,

13.2 VARIABLES

The internal CNC variables which can be accessed by high-level language are grouped in tables and can be read-only or read-write variables.

There is a group of mnemonics for showing the different fields of the table of variables. In this way, if it is required to access an element from one of these tables, the required field will be indicated by means of the corresponding mnemonic (for example TOR) and then the required element (TOR3).

The variables available at the CNC can be classified in the following way:

- General purpose parameters or variables
- Variables associated with tools.
- Variables associated with zero offsets.
- Variables associated with machine parameters
- Variables associated with work zones
- Variables associated with feedrates
- Variables associated with position coordinates
- Variables associated with the spindle
- Variables associated with the PLC
- Variables associated with local parameters
- Other variables

Variables which access to real values of the CNC interrupt the preparation of blocks and the CNC waits for each command to be performed before restarting block preparation.

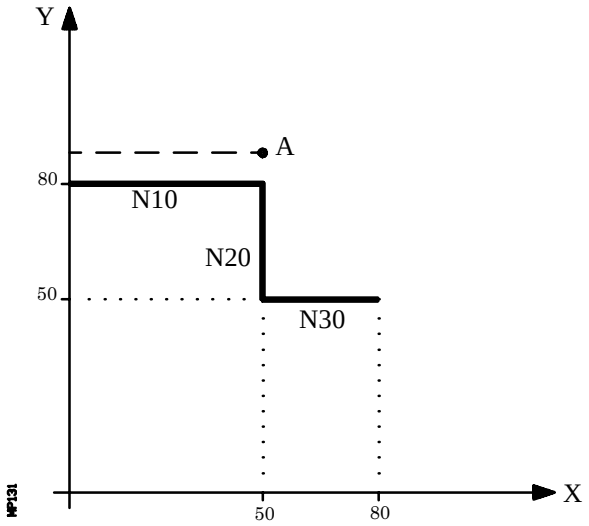
Thus, precaution must be taken when using this type of variable, as should they be placed between machining blocks which are working with compensation, undesired profiles may be obtained.

Example:

The following program blocks are performed in a section with G41 compensation.

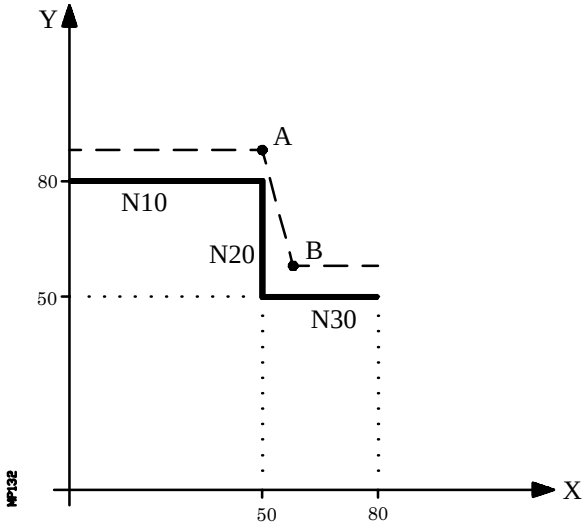
```
.......
.......
N10 X50 Y80
N15 (P100=POSX);Assigns the value of the real coordinate in X to parameter P100
N20 X50 Y590
N30 X80 Y50
.......
.......
```

Block N15 interrupts block preparation and the execution of block N10 will finish at point A.



Once the execution of block N15 has ended, the CNC will continue block preparation from block N20 on.

As the next point corresponding to the compensated path is point “B”, the CNC will move the tool to this point, executing path “A-B”.



As can be observed, the resulting path is not the desired one, and therefore it is recommended to avoid the use of this type of variable in sections requiring tool compensation.

13.2.1 GENERAL PURPOSE PARAMETERS OR VARIABLES

The CNC has two types of general purpose variables: local parameters P0-P25 and global parameters P100-P299.

Programmers may use general purpose variables when editing their own programs. Later and during execution, the CNC will replace these variables with the values assigned to it at that time.

Example:

GP0 XP1 Y100	—> G1 X-12.5 Y100
(IF(P100*P101 EQ P102)GOTO N100)	—> (IF(2*5 EQ 12)GOTO N100)

The use of these global purpose variables will depend on the type of block in which they are programmed and the channel of execution.

In block programmed in ISO code parameters can be associated with all fields, G X..C F S T D M. The block label number will be defined with a numerical value.

If parameters are used in blocks programmed in high-level language, these can be programmed within any expression.

Programmes which are executed in the user channel may contain any global parameter, but may not use local parameters.

The CNC will update the parameter table after processing the operations indicated in the block which is in preparation. This operation is always done before executing the block and for this reason, the values shown in the table do not necessarily have to correspond to the block being executed.

If the Execution Mode is abandoned after interrupting the execution of the program, the CNC will update the parameter tables with values corresponding to the block which was being executed.

When accessing the local parameter and global parameter table, the value assigned to each parameter may be expressed in decimal notation (4127.423) or in scientific notation (=23476 E-3).

This CNC has high level statements which allow the definition and use of subroutines which can be called from the main program, or from another subroutine, it also being possible to call a second subroutine, from the second to a third, etc. The CNC limits these calls, allowing up to a maximum of 15 nesting levels.

26 local parameters (P0-P25) can be assigned to a subroutine. These parameters which will be unknown for blocks external to the subroutine may be referenced by the blocks of this subroutine.

The CNC allows local parameters to be assigned to more than one subroutine, 6 nesting levels of local parameters being possible, within the 15 nesting levels of a subroutine.

Local parameters used in high-level language may be defined either using the above format or by using the letter A-Z, except for N, so that A is equal to P0 and Z to P25.

The following example shows these two methods of definition:

```
(IF((P0+P1)   *  P2/P3   EQ  P4)   GOTO N100)
(IF((A+B)     *  C/D      EQ  E)    GOTO N100)
```

When using a parameter name (letter) for assigning a value to it (A instead of P0, for example), if the arithmetic expression is a constant, the statement can be abbreviated as follows:

(P0 = 13.7) —> (A = 13.7) —> (A13.7)

Be careful when using parenthesis since M30 is not the same as (M30). The CNC interprets (M30) as a high level statement meaning (P12 = 30) and not the execution of the miscellaneous M30 function.

The global parameter (P100-P299) can be used throughout the program by any block, irrespective of the nesting level.

Multiple machining (G60, G61, G62, G63, G64, G65) and machining canned cycles (G69, G81 ... G89) use a local parameter nesting level when active.

Machining canned cycles use the global parameter P299 for internal calculations and probing canned cycles use global parameters P294 to P299.

13.2.2 VARIABLES ASSOCIATED WITH TOOLS

These variables are associated with the tool offset table, tool table and tool magazine table, so the values which are assigned to or read from these fields will comply with the formats established for these tables.

Tool offset table:

R,L,I,K	They are given in the active units: If G70, in inches. Max.: ±3937.00787 If G71, in millimeters. Max.: ±99999.9999 If rotary axis in degrees. Max.: ±99999.9999

Tool table

Tool offset number	0...NT OFFSET (maximum 255)
Family code	If normal tool $0 \leq n < 200$ If special tool $200 \leq n \leq 255$
Nominal life	0...65535 minutes or operations.
Real life	0.99999.99 minutes or 99999 operations

Tool magazine table

Contents of each magazine position

Tool number 1 ...NTOOL
(maximum 255)
0 Empty
-1 Cancelled

Position of tool in magazine

Position number 1 ..NPOCKET
(maximum 255)
0 On spindle
-1 Not found
-2 In change position

Read-only variables

TOOL: Returns the active tool number
(P100 = TOOL); assigns the number of the active tool to P100

TOD: Returns the active tool offset number

NXTOOL: Returns the next tool number, selected but is awaiting the execution of M06 to be active.

NXTOD: Returns the number of the tool offset corresponding to the next tool, selected but is awaiting the execution of M06 to be active.

TMZPn: Returns the position occupied in the tool magazine by the indicated tool (n).

Read-write variables

TORn: This variable allows the value assigned to the Radius of the indicated tool offset (n) on the tool offset table to be read or modified.

(P110 = TOR3); Assigns the R value of tool offset 3 to Parameter 3).
(TOR3 = P111); Assigns the value of parameter P111 to R of tool offset 3)

TOLn: This variable allows the value assigned to the Length of the indicated tool offset (n) to be read or modified on the tool offset table.

TOIn: This variable allows the value assigned to the radius wear (I) of the indicated tool offset (n) to be read or modified on the tool offset table.

TOKn: This variable allows the value assigned to the length wear (K) of the indicated tool offset (n) to be read or modified on the tool offset table.

TLFDn: This variable allows the tool offset number of the indicated tool (n) to be read or modified on the tool table.

TLFFn: This variable allows the family code of the indicated tool (n) to be read or modified on the tool table.

TLFNn: This variable allows the value assigned as the nominal life of the indicated tool (n) to be read or modified on the tool table.

TLFRn: This variable allows the value corresponding to the real life of the indicated tool (n) to be read or modified on the tool table.

TMZTn: This variable allows the contents of the indicated position (n) to be read or modified on the tool magazine table.

13.2.3 VARIABLES ASSOCIATED WITH ZERO OFFSETS

These variables are associated with the zero offsets and may correspond to the table values or to those currently preset either by means of function G92 or manually in the JOG mode.

The zero offsets which are possible in addition to the additive offset indicated by the PLC, are G54, G55, G56, G57, G58 and G59.

The values for each axis are given in the active units:

If G70, in inches. Max.: ±3937.00787
If G71, in millimeters. Max.: ±99999.9999
If rotary axis in degrees. Max.: ±99999.9999

Although there are variables which refer to each axis, the CNC only allows those referring to the selected axes in the CNC. Thus, if the CNC controls axes X, Y, Z, U and B, it only allows the variables ORGX, ORGY, ORGZ,. ORGU and ORGB in the case of ORG(X-C).

Read-only variables

ORG(X-C) Returns the value of the active zero offset in the selected axis. The value of the additive offset indicated by the PLC is not included in this value.

(P100 = ORGX); assigns to P100 the X value of the part zero active for the X axis. This value could have been set either by means of function G92 or by the variable "**ORG(X-C)n**".

PORGF: Returns the abscissa value of the polar coordinate origin with respect to the Cartesian origin.

PORGS: Returns the ordinate value of the polar coordinate origin with respect to the cartesian origin.

Read-write variables

ORG(X-C)n: This variable allows the value of the selected axis to be read or modified on the table corresponding to the indicated zero offset (n).

(P110 = ORGX55); Assigns the value of X to parameter P110 on the table corresponding to zero offset G55.

(ORGY 54 = P111); Assigns the value of parameter P111 to the Y axis on the table corresponding to G54 zero offset.

PLCOF(X-C) This variable allows the value of the selected axis to be read or modified on the additive zero offset table indicated by the PLC.

If any of the PLCOF(X-C) variables are accessed, **block preparation is interrupted** and the CNC waits for this command to be executed to begin block preparation again.

13.2.4 VARIABLES ASSOCIATED WITH FUNCTION G49

With function G49, it is possible to define a coordinate transformation or, in other words, the incline plane resulting from that transformation.

Read-only variables associated with the definition of function G49:

ORGROX X coordinate of the new part zero referred to home.
ORGROY Y coordinate of the new part zero referred to home.
ORGROZ Z coordinate of the new part zero referred to home.

ORGROA Value assigned to parameter A
ORGROB Value assigned to parameter B
ORGROC Value assigned to parameter C

ORGROI Value assigned to parameter I
ORGROJ Value assigned to parameter J
ORGROK Value assigned to parameter K

ORGROQ Value assigned to parameter Q
ORGROR Value assigned to parameter R
ORGROS Value assigned to parameter S

GTRATY Type of G49 programmed 0 = no G49 programmed
1= G49 X Y Z A B C 2= G49 X Y Z Q R S
3= G49 T X Y Z S 4= G49 X Y Z I J K R S

Every time G49 is programmed, the CNC updates the values of the parameters that have be defined.

For example, when programming G49 XYZ ABC
The CNC updates the following variables
ORGROX, ORGROY, ORGROZ
ORGROA, ORGROB, ORGROC
The rest of the variables keep their previous values.

Read-Write variables updated by the CNC once function G49 is executed:

When having a swivel or angled spindle, machine parameter "XFORM (P93) with a value of 2 or 3, the CNC shows the following information:

TOOROF Indicates the position to be occupied by the main rotary axis of the spindle in order for the tool to be positioned perpendicular to the indicated incline plane.

TOOROS Indicates the position to be occupied by the secondary rotary axis of the spindle in order for the tool to be positioned perpendicular to the indicated incline plane.

By accessing variable TOOROF or TOOROS, the CNC interrupts block preparation and waits for that command to be executed before resuming block preparation.

13.2.5 VARIABLES ASSOCIATED WITH MACHINE PARAMETERS

Variables associated with machine parameters are read-only variables.

In order to become familiar with the values returned it is advisable to consult the installation and start-up manual.

Values 1/0 correspond to the parameters which are defined with YES/NO, +/- and ON/OFF.

The coordinate and feedrate values are given in the active units:

If G70, in inches. Max.: ±3937.00787
If G71, in millimeters. Max.: ±99999.9999
If rotary axis in degrees. Max.: ±99999.9999

Read-only variables

MPGn: Returns the value assigned to the general machine parameter (n).

(P110=MPG 8);assigns the value of the general machine parameter "INCHES" to parameter P110, if millimeters P110=0 and if inches P110=1.

MP(X-C)n Returns the value which was assigned to the machine parameter (n) of the indicated axes.

(P110=MPY 1); assigns the value of the machine parameter P1 to arithmetic parameter P110 of the Y axis "DFORMAT", which indicates the format used in its display.

MPSn: Returns the value which was assigned to the main spindle machine parameter (n).

MPSSn: Returns the value which was assigned to the secondary spindle machine parameter (n).

MPASn: Returns the value of the machine parameter (n) for the auxiliary spindle.

MPLCn: Returns the value which was assigned to the PLC machine parameter (n)

13.2.6 VARIABLES ASSOCIATED WITH WORK ZONES

Variables associated with work zones are read-only variables.

The values of the limits are given in the active units:

If G70, in inches. Max.: ±3937.00787
If G71, in millimeters. Max.: ±99999.9999
If rotary axis in degrees. Max.: ±99999.9999

The status of the work zones is determined according to the following code:

0 = Disabled.
1 = Enabled as no-entry zone.
2 = Enabled as no-exit zone.

Read-only variables

FZONE: Returns the status of work zone 1.
(P100=FZONE); assigns to parameter P100 the status of work zone 1.

FZLO(X-C) Returns the value of the lower limit of Zone 1 according to the selected axis (X-C).

FZUP(X-C) Returns the value of the upper limit of Zone 1 according to the selected axis (X-C).

SZONE: Returns the status of work zone 2.

SZLO(X-C) Returns the value of the lower limit of Zone 2 according to the selected axis (X-C).

SZUP(X-C) Returns the value of the upper limit of Zone 2 according to the selected axis (X-C).

TZONE: Returns the status of work zone 3.

TZLO(X-C) Returns the value of the lower limit of Zone 3 according to the selected axis (X-C).

TZUP(X-C) Returns the value of the upper limit of Zone 3 according to the selected axis (X-C).

FOZONE: Returns the status of work zone 4.

FOZLO(X-C) Returns the value of the lower limit of Zone 4 according to the selected axis (X-C).

FOZUP(X-C) Returns the value of the upper limit of Zone 4 according to the selected axis (X-C).

13.2.7 VARIABLES ASSOCIATED WITH FEEDRATES

Read-only variables associated with the actual feedrate

FREAL: Returns the real feedrate of the CNC in mm/min. or inches/min.

(P100 = FREAL); Assigns the real feedrate value of the CNC to parameter P100

Read-only variables associated with function G49

FEED: Returns the feedrate selected in the CNC by means of the G94 function. This will be in mm/minute or inches/minute.

This feedrate can be indicated by program, by the PLC or DNC, and the CNC selects one of these, the one with the highest priority being that indicated by DNC and the one with the lowest priority that indicated by program.

DNCF: Returns the feedrate, in mm/minute or inches/minute, selected by DNC. If this has a value of 0 it means that it is not selected.

PLCF: Returns the feedrate, in mm/minute or inches/minute, selected by PLC. If this has a value of 0 it means that it is not selected.

PRGF: Returns the feedrate, in mm/minute or inches/minute, selected by program.

Read-only variables associated with function G95

FPREV: Returns the feedrate selected in the CNC by means of the G95 function. This will be in mm/rev. or inches/rev.

This advance can be indicated by program, by the PLC or DNC, and the CNC selects one of these, the one with the highest priority being that indicated by DNC and the one with the lowest priority that indicated by program.

DNCFPR: Returns the feedrate, in mm/rev. or inches/rev., selected by DNC. If this has a value of 0 it means that it is not selected.

PLCFPR: Returns the feedrate, in mm/rev. or inches/rev., selected by PLC. If this has a value of 0 it means that it is not selected.

Read-only variables associated with function G32

PRGFIN: Returns the feedrate, in 1/min selected by program.

Also, the CNC variable FEED associated with G94 will show the resulting feedrate in mm/min or inches/min.

Read-only variables associated with Feedrate Override

PRGFPR: Returns the feedrate, in mm/rev. or inches/rev., selected by program.

FRO: Returns the Feedrate Override (%) selected at the CNC. This will be given by an integer between 0 and "MAXFOVR" (maximum 255).

This feedrate percentage may be indicated by the PLC, by DNC or from the front panel, and the CNC will select one of them, the order of priority (from highest to lowest) being: by program, by DNC, by PLC and from the switch.

DNCFRO: Returns the Feedrate Override % selected by DNC. If this has a value of 0 it means that it is not selected.

PLCFRO: Returns the Feedrate Override % selected by PLC. If this has a value of 0 it means that it is not selected.

CNCFR0: Returns the Feedrate Override % selected from the switch at the CNC Operator Panel.

Read-write variables

PRGFRO: This variable allows the feedrate percentage selected by program to be read or modified. This will be given by an integer between 0 and "MAXFOVR" (maximum 255). If it has a value of 0 this means that it is not selected.

(P110 = PRGFRO); assigns to P110 the % of feedrate override selected by program

(PFRGFRO = P111); sets the feedrate override % selected by program to the value of P111.

13.2.8 VARIABLES ASSOCIATED WITH COORDINATES

The coordinate values for each axis are given in the active units:

If G70, in inches. Max.: ±3937.00787
If G71, in millimeters. Max.: ±99999.9999
If rotary axis in degrees. Max.: ±99999.9999

Read-only variables

PPOS(X-C) Returns the programmed theoretical coordinate of the selected axis.

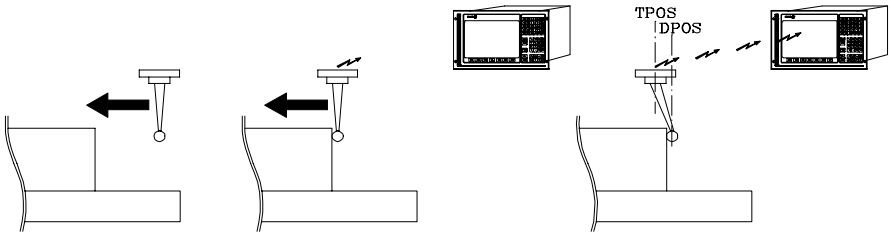
(P100) = PPOSX); assigns to P100 the programmed theoretical position of the X axis.

POS(X-C) Returns the real coordinate of the selected axis referred to machine reference zero (home).

TPOS(X-C) Returns the theoretical coordinate (real + following error) of the selected axis referred to machine reference zero (home).

DPOS(X-C) The CNC updates this variable whenever probing operations are carried out, same as with G75, G76 functions and probing cycles (Probe, Digit).

When the digital probe and the CNC communicate with each other via infrared beams, there could be a delay of a few milliseconds from when the probe touches the part until the moment the CNC receives the probe signal.



Although the probe keeps moving until the CNC receives the probe signal, the CNC assumes the value assigned to general machine parameter PRODEL and provides the following information (variables associated with coordinates):

TPOS Actual position of the probe when the CNC receives the probe signal.
DPOS Theoretical position of the probe when it touched the part.

FLWE(X-C) Returns the amount of following error of the selected axis.

DEFLEX
DEFLEY
DEFLEZ: They return the current deflection of the Renishaw probe SP2 along each axis, **X, Y, Z.**

When accessing one of these variables (POS(X-C), TPOS(X-C), DPOS(X-C), FLWE(X-C), DEFLEX, DEFLEY or DEFLEZ), **block preparation is interrupted** and the CNC waits for that command to be executed before resuming block preparation.

Read-write variables

DIST(X-C) These variables allow the distance travelled by the selected axis to be read or modified. This value is accumulative and it is very useful when it is required to perform an operation which depends on the distance travelled by the axes, for example: in their lubrication.

(P100= DISTX); assigns to P100 the distance travelled by the X axis

(DISTZ = P111); presets the variable indicating the distance travelled by the Z axis with the value of arithmetic parameter P111.

If any of the DIST(X-C) variables are accessed, **block preparation is interrupted** and the CNC waits for this command to be executed to resume block preparation.

LIMPL(X-C):
LIMMI(X-C): With these variables, it is possible to set a second travel limit for each axis, LIMPL for the upper limit and LIMMI for the lower limit.

Since the second limits are activated or deactivated from the PLC, through general logic input ACTLIM2 (M5052), besides setting the limits, an auxiliary M code must be executed to let it know.

It is also recommended to execute function G4 after the change so the CNC executes the following blocks with the new limits.

The second travel limit will be taken into consideration when the first one has been set using axis machine parameters LIMIT+ (P5) and LIMIT- (P6).

13.2.9 VARIABLES ASSOCIATED WITH THE ELECTRONIC HANDWHEELS

Read-only variables

HANPF
HANPS
HANPT
HANPFO They return the number of pulses of the first (HANPF), second (HANPS), third (HANPT) or fourth (HANPFO) handwheel received since the CNC was turned on. Regardless of whether the handwheel is connected to the AXES module or to the I/O module.

13.2.10 VARIABLES ASSOCIATED WITH THE MAIN SPINDLE

In these variables associated with the spindle, their values are given in revolutions per minute and the main spindle override values are given in integers from 0 to 255.

Read-only variables

SREAL: Returns the real main spindle turning speed in revolutions per minute.

(P100 = SREAL); assigns to P100 the real turning speed of the main spindle.

If this variable is accessed, **block preparation is interrupted** and the CNC waits for this command to be executed to resume block preparation.

SPEED: Returns, in revolutions per minute, the main spindle speed selected at the CNC.

This turning speed can be indicated by program, by the PLC or DNC, and the CNC selects one of these, the one with the highest priority being that indicated by DNC and the one with the lowest priority that indicated by program.

DNCS: Returns the turning speed in revolutions per minute, selected by DNC. If this has a value of 0 it means that it is not selected.

PLCS: Returns the turning speed in revolutions per minute selected by PLC. If this has a value of 0 it means that it is not selected.

PRGS: Returns the turning speed in revolutions per minute, selected by program.

SSO: Returns the Override (%) of the main spindle speed selected at the CNC. This will be given by an integer between 0 and "MAXSOVR" (maximum 255).

This spindle speed percentage may be indicated by the PLC, by DNC or from the front panel, and the CNC will select one of them, the order of priority (from highest to lowest) being: by program, by DNC, by PLC and from the front panel.

DNCSSO: Returns the main spindle speed percentage selected by DNC. If this has a value of 0 it means that it is not selected.

PLCSSO: Returns the main spindle speed percentage selected by PLC. If this has a value of 0 it means that it is not selected.

CNCSSO: Returns the main spindle speed percentage selected from the front panel.

SLIMIT: Returns, in revolutions per minute, the value established for the main spindle speed limit selected at the CNC.

This limit can be indicated by program, by the PLC or DNC, and the CNC selects one of these, the one with the highest priority being that indicated by DNC and the one with the lowest priority that indicated by program.

DNCSL: Returns the main spindle speed limit in revolutions per minute, selected by DNC. If this has a value of 0 it means that it is not selected.

PLCSL: Returns the main spindle speed limit in revolutions per minute selected by PLC. If this has a value of 0 it means that it is not selected.

PRGSL: Returns the main spindle speed limit in revolutions per minute, selected by program.

POSS: Returns the main spindle real position value, when it is in closed loop (M19). Its value will be given in 0.0001 degree units between ±999999999.

RPOSS: Returns the main spindle real position value. Its value will be given in 0.0001 degree units between 0 and 360º,

TPOSS: Returns the main spindle theoretical position value. Its value will be given in 0.0001 degree units between ±999999999.

RTPOSS: Returns the main spindle theoretical position value. Its value will be given in 0.0001 degree units between 0 and 360º.

FLWES: Returns the spindle following error when it is operating in closed loop (M19).

When accessing one of these variables (POSS, RPOSS, TPOSS RTPOSS or FLWES), **block preparation is interrupted** and the CNC waits for that command to be executed before resuming block preparation.

Read-write variables

PRGSSO: This variable allows the percentage of the main spindle speed selected by program to be read or modified. This will be given by an integer between 0 and "MAXSOVR" (maximum 255). If this has a value of 0 it means that it is not selected.

(P110 = PRGSSO); assigns to P110 the % of the main spindle speed selected by program.

(PRGSSO = P111); sets the value indicating the main spindle speed % seleceted by program to the value of arithmetic parameter P111.

*13.2.11 VARIABLES ASSOCIATED WITH THE 2ND SPINDL*E

In these variables associated with the spindle, their values are given in revolutions per minute and the 2nd spindle override values are given in integers from 0 to 255.

Read-only variables

SSREAL: Returns the real 2nd spindle turning speed in revolutions per minute.

(P100 = SRSEAL); assigns to P100 the real turning speed of the 2nd spindle.

If this variable is accessed, **block preparation is interrupted** and the CNC waits for this command to be executed to resume block preparation.

SSPEED: Returns, in revolutions per minute, the 2nd spindle speed selected at the CNC.

This turning speed can be indicated by program, by the PLC or DNC, and the CNC selects one of these, the one with the highest priority being that indicated by DNC and the one with the lowest priority that indicated by program.

SDNCS: Returns the turning speed in revolutions per minute, selected by DNC. If this has a value of 0 it means that it is not selected.

SPLCS: Returns the turning speed in revolutions per minute selected by PLC. If this has a value of 0 it means that it is not selected.

SPRGS: Returns the turning speed in revolutions per minute, selected by program.

SSSO: Returns the Override (%) of the 2nd spindle speed selected at the CNC. This will be given by an integer between 0 and "MAXSOVR" (maximum 255).

This spindle speed percentage may be indicated by the PLC, by DNC or from the front panel, and the CNC will select one of them, the order of priority (from highest to lowest) being: by program, by DNC, by PLC and from the front panel.

SDNCSO: Returns the 2nd spindle speed percentage selected by DNC. If this has a value of 0 it means that it is not selected.

SPLCSO: Returns the 2nd spindle speed percentage selected by PLC. If this has a value of 0 it means that it is not selected.

SCNCSO: Returns the 2nd spindle speed percentage selected from the front panel.

SSLIMI: Returns, in revolutions per minute, the value established for the 2nd spindle speed limit selected at the CNC.

This limit can be indicated by program, by the PLC or DNC, and the CNC selects one of these, the one with the highest priority being that indicated by DNC and the one with the lowest priority that indicated by program.

SDNCSL: Returns the 2nd spindle speed limit in revolutions per minute, selected by DNC. If this has a value of 0 it means that it is not selected.

SPLCSL: Returns the 2nd spindle speed limit in revolutions per minute selected by PLC. If this has a value of 0 it means that it is not selected.

SPRGSL: Returns the 2nd spindle speed limit in revolutions per minute, selected by program.

SPOSS: Returns the 2nd spindle real position value, when it is in closed loop (M19). Its value will be given in 0.0001 degree units between ±999999999.

SRPOSS: Returns the 2nd spindle real position value. Its value will be given in 0.0001 degree units between 0 and 360º,

TPOSS: Returns the 2nd spindle theoretical position value. Its value will be given in 0.0001 degree units between ±999999999.

SRTPOS: Returns the 2nd spindle theoretical position value. Its value will be given in 0.0001 degree units between 0 and 360º.

SFLWES: Returns the spindle following error when it is operating in closed loop (M19).

When accessing one of these variables (SPOSS, SRPOS, STPOSS, SRTPOS or SFLWES), **block preparation is interrupted** and the CNC waits for that command to be executed before resuming block preparation.

Read-write variables

SPRGSO: This variable allows the percentage of the 2nd spindle speed selected by program to be read or modified. This will be given by an integer between 0 and "MAXSOVR" (maximum 255). If this has a value of 0 it means that it is not selected.

(P110 = SPRGSO); assigns to P110 the % of the 2nd spindle speed selected by program.

(SPRGSO = P111); sets the value indicating the 2nd spindle speed % seleceted by program to the value of arithmetic parameter P111.

13.2.12 VARIABLES ASSOCIATED WITH THE PLC

It should be borne in mind that the PLC has the following resources:

Inputs	(I1 thru I256)
Outputs	(O1 thru O256).
Marks	M1 thru M5957)
Registers	(R1 thru R499) of 32 bits each.
Timers	(T1 thru T256) with a timer count in 32 bits.
Counters	(C1 thru C256) with a counter count in 32 bits.

If any variable is accessed which allows the status of a PLC variable to be read or modified (I,O,M,R,T,C), **block preparation is interrupted** and the CNC waits for this command to be executed in order to restart block preparation.

Read-only variables

PLCMSG: Returns the number of the active PLC message with the highest priority and will coincide with the number displayed on screen (1...128). If there is none, it returns 0.

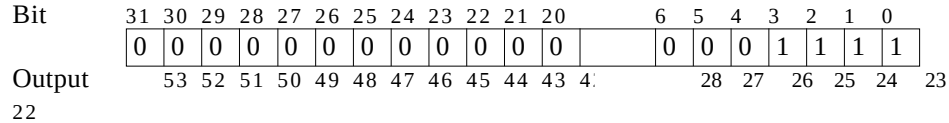
(P100 = PLCMSG); assigns to P100 the number of the active PLC message with the highest priority.

Read-write variables

PLCIn: This variable allows 32 PLC inputs to be read or modified starting with the one indicated (n).

The value of the inputs which are used by the electrical cabinet cannot be modified as their values are determined by it. Nevertheless, the status of the remaining inputs can be modified.

PLCOn: This variable allows 32 PLC outputs to be read or modified starting from the one indicated (n).



PLCMn: This variable allows 32 PLC marks to be read or modified starting from the one indicated (n).

PLCRn: This variable allows the status of 32 register bits to be read or modified starting from the one indicated (n).

PLCTn: This variable allows the timer count to be read or modified starting from the one indicated (n).

PLCCn: This variable allows the counter count to be read or modified starting from the one indicated (n).

13.2.13 VARIABLES ASSOCIATED WITH LOCAL PARAMETERS

The CNC allows 26 local parameters (P0-P25) to be assigned to a subroutine, by using mnemonics PCALL and MCALL.

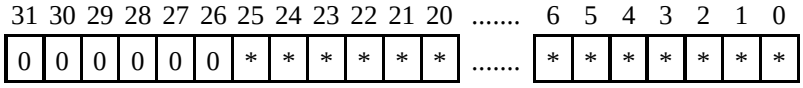
In addition to performing the required subroutine these mnemonics allow local parameters to be initialized.

Read-only variables

CALLP: Allows us to know which local parameters have been defined and which have not, in the call to the subroutine by means of the PCALL or MCALL mnemonic.

The information will be given in the 26 least significant bits (bits 0..25), each of these corresponding to the local parameter of the same number, as well as bit 12 corresponding to P12.

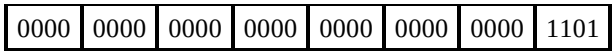
Each bit will indicate if the corresponding local parameter has been defined (=1) or not (=0).



Example:

```
(PCALL 20, P0=20, P2=3, P3=5)      ;Call to subroutine 20.
....
....
(SUB 20)                           ;Beginning of subroutine 20
P100=CALLP)
....
....
```

In parameter P100 the following will be obtained:



LSB

13.2.14 SERCOS VARIABLES

They are used for data exchange between the CNC and the servo drives via Sercos interface.

Read-only variables

TSVAR(X-C) identifier ..... for the axes
TSVARS identifier ..... for the main spindle
TSSVAR identifier ..... for the second spindle

It returns the third attribute of the sercos variable corresponding to the "identifier". The third attribute is used in particular software applications and its information is coded according to the Sercos standard.

(P110=SVARX 40) assigns to parameter P110 the third attribute of the sercos variable of identifier 40 of the X axis which corresponds to "VelocityFeedback"

Write-only variables

SETGE(X-C) ..... for the axes
SETGES ..... for the main spindle
SSETGS ..... for the second spindle

The drive may have up to 8 work ranges or gears (0 through 7). Sercos identifier 218, GearRatioPreselection.

It may also have up to 8 parameter sets (0 through 7). Sercos indentifier 217, ParameterSetPreselection.

These variables permit changing the work range (gear) or the paramete set for each drive.

The 4 least significant bits of these variables must indicate the work gear and the 4 most significant bits the parameter set to be selected.

Read-Write variables

SVAR(X-C) identifier ..... for the axes
SVARS identifier ..... for the main spindle
SSVAR identifier ..... for the second spindle

They permit reading or modifying the value of the sercos variable corresponding to the axis identifier.

(P110=SVARX 40) assigns to parameter P110 the value of the sercos variable of identifier 40 of the X axis which corresponds to the "VelocityFeedback"

13.2.15 OTHER VARIABLES

Read-only variables

OPMODE: Returns the code corresponding to the selected operating Mode.

0 = Main menu.

10 = Automatic execution.
11 = Single block execution.
12 = MDI in EXECUTION
13 = Tool inspection

20 = Theoretical path movement simulation
21 = G functions simulation
22 = G, M, S and T functions simulation
23 = Simulation with movement on main plane
24 = Simulation with rapid movement

30 = Normal editing
31 = User editing
32 = TEACH-IN editing
33 = Interactive editor
34 = Profile editor

40 = Movement in continuous JOG
41 = Movement in incremental JOG
42 = Movement with electronic handwheel
43 = HOME search in JOG
44 = Position preset in JOG
45 = Tool calibration
46 = MDI in JOG
47 = JOG user operation

50 = Zero offset table
51 = Tool Offset table
52 = Tool table
53 = Tool magazine table
54 = Global parameter table
55 = Local parameter table

60 = Utilities

70 = DNC status
71 = CNC status

80 = Editing PLC files
81 = Compiling PLC program
82 = PLC monitoring
83 = Active PLC messages
84 = Active PLC pages
85 = Save PLC program
86 = Restore PLC program
87 = “PLC resources in use” mode
88 = PLC statistics

90 = Graphic Editor

100 = General machine parameter table
101 = Axis machine parameter tables
102 = Spindle machine parameter tables
103 = Serial port machine parameter tables
104 = PLC machine parameter table
105 = M function table
106 = Spindle and cross compensation table
107 = Machine parameter table for Ethernet

110 = Diagnosis: configuration
111 = Diagnosis: hardware test
112 = Diagnosis: RAM memory test
113 = Diagnosis: FLASH memory test
114 = User diagnosis
115 = Hard Disk diagnosis (HD)

OPMODA Indicates the operating mode currently selected when working with the main channel.

Use the OPMODE variable to know at any time the selected operating mode (main channel, user channel, PLC channel).

This information is given at the least significant bits with a "1" when active and with a "0" when not active or when it is not available in the current version.

bit 0 Program in execution.
bit 1 Program in simulation.
bit 2 Block in execution via MDI, JOG
bit 3 Repositioning in progress.
bit 4 Program interrupted, by CYCLE STOP
bit 5 MDI, JOG Block interrupted
bit 6 Repositioning interrupted
bit 7 In tool inspection
bit 8 Block in execution via CNCEX1
bit 9 Block via CNCEX1 interrupted
bit 10 CNC ready to accept JOG movements: jog, handwheel, teach-in, inspection.
bit 11 CNC ready to receive the CYCLE START command: execution, simulation and MDI modes.
bit 12 The CNC is not ready to execute anything involving axis or spindle movement.

OPMODB Indicates the type of simulation currently selected. This information is given at the least significant bits with a "1" indicating the currently selected one.

bit 0 Theoretical path
bit 1 G functions
bit 2 G M S T functions
bit 3 Main plane
bit 4 Rapid

OPMODC Indicates the axes selected by Handwheel. This information is given at the least significant bits indicating with a "1" the one currently selected.

bit 8	bit 7	bit 6	bit 5	bit 4	bit 3	bit	bit 1	bit 0
		Axis 7	Axis 6	Axis 5	Axis 4	Axis 3	Axis 2	Axis 1

The axis number corresponds to the order it is programmed.
Example: If the CNC controls the X, Y, Z, U, B, C axes, Axis 1 will be the X axis, Axis 2= Y, Axis 3=Z, Axis 4= U, Axis 5= B, Axis 6= C.

NBTOOL Indicates the tool number being managed.

Example: There is a manual tool changer. Tool T1 is currently selected and the operator requests tool T5.

The subroutine associated with the tools may contain the following instructions:

(P103 = NBTOOL)
(MSG “SELECT T?P103 AND PRESS CYCLE START”)

Instruction (P103 = NBTOOL) assigns the number of the tool currently being managed to parameter P103. Therefore, P103=5

The message displayed by the CNC will be “”SELECT T5 AND PRESS CYCLE START”**.**

PRGN: Returns the program number being executed. Should none be selected, a value of -1 is returned.

BLKN: Returns the label number of the last block executed.

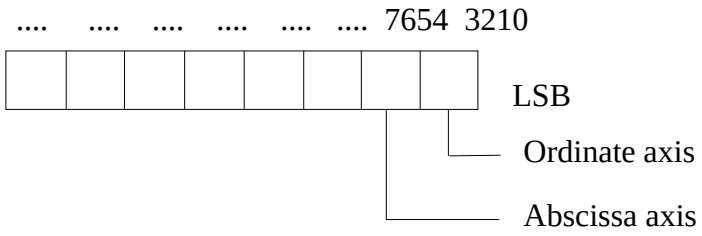
GSn: Returns the status of the G function indicated (n). 1 if it is active and 0 if not.

(P120=GS17); assigns the value 1 to parameter P120 if the G17 function is active and 0 if not.

MSn: Returns the status of the M function indicated (n). 1 if it is active and 0 if not.

This variable provides the status of M00, M01, M02, M03, M04, M05, M06, M08, M09, M19, M30, M41, M42, M43, M44 and M45 functions.

PLANE: Returns data on the abscissa axis (bits 4 to 7) and the ordinate axis (bits 0 to 3) of the active plane in 32 bits and in binary.



The axes are coded in 4 bits and indicate the axis number (from 1 to 6) according to the programming order.

Example: If the CNC controls the X,Y,Z,U,B,C axes and is selected in the ZX plane (G18).

(P122 = PLANE) assigns value $31 to parameter P122.

0000	0000	0000	0000	0000	0000	0011	0001

LSB

LONGAX: Returns the number (1 to 6) according to the programming order corresponding to the longitudinal axis. This will be the one selected with the G15 function and, by default, the axis perpendicular to the active plane, if this is XY, ZX or YZ.

Example: If the CNC controls the X, Y, Z, U, B,C axes and the U axis is selected.

(P122 = LONGAX) assigns the value 4 to parameter 122.

MIRROR Returns in the least significant bits in a group of 32 bits, the status of the mirror image of each axis, 1 in the case of being active and 0 if not.

bit 8	bit 7	bit 6	bit 5	bit 4	bit 3	bit	bit 1	bit 0
		Axis 7	Axis 6	Axis 5	Axis 4	Axis 3	Axis 2	Axis 1

The name of the axis corresponds to the number according to their programming order.

Example: If the CNC controls axes X, Y, Z, U, B, C
Axis 1=X, Axis2=Y, Axis3=Z, Axis4=U, Axis5=B, Axis6=C.

SCALE: Returns the general scaling factor applied.

SCALE(X-C): Returns the specific scaling factor of the axis indicated (X-C).

ORGROT: Returns the turning angle of the coordinate system selected with the G73 function. Its value is given in degrees. Max. ±99999.9999º

ROTPF: Returns the abscissa value of the rotation center with respect to the cartesian coordinate origin. It is given in the active units:

If G70, in inches. Max. ±3937.00787
If G71, in millimeters. Max. ±99999.9999

ROTPS: Returns the ordinate value of the rotation center with respect to the cartesian coordinate origin. It is given in the active units:

If G70, in inches. Max. ±3937.00787
If G71, in millimeters. Max. ±99999.9999

PRBST: Returns the status of the probe.

0 = The probe is **not** touching the part.
1 = The probe is touching the part.

CLOCK: Returns in seconds the time indicated by the system clock. Possible values 0...4294967295

If this variable is accessed, **block preparation is interrupted** and the CNC waits for this command to be executed to resume block preparation.

TIME: Returns the time in hours-minutes-seconds format.

(P150=TIME); assigns hh-mm-ss to P150. For example if the time is 18h 22m 34 sec., P150 will contain 182234.

If this variable is accessed, **block preparation is interrupted** and the CNC waits for this command to be executed to resume block preparation.

DATE: Returns the date in year-month-day format.

(P151=DATE); assigns year-month-day to P151. For example if the date is April 25th 1992, P151 will contain 920425.

If this variable is accessed, **block preparation is interrupted** and the CNC waits for this command to be executed to resume block preparation.

CYTIME: Returns in hundredths of a second the time it has taken to make the part. Possible values 0...4294967295

If this variable is accessed, **block preparation is interrupted** and the CNC waits for this command to be executed to resume block preparation.

FIRST: Indicates whether it is the first time that a program has been run. It returns a value of 1 if it is the first time and 0 for the remainder of times.

A first-time execution is considered as being one made:

After turning on the CNC.
After pressing the "Shift-Reset" keys.
Every time a new program is selected.

ANAIn: Returns in volts and in ±1.4 format (values ±5 Volts), the status of the analog input indicated (n), it being possible to select one among eight (1...8) analog inputs.

If this variable is accessed, **block preparation is interrupted** and the CNC waits for this command to be executed to resume block preparation.

AXICOM Returns in the 3 least significant bytes the axis pairs toggled with function G28.

		Pair 3		Pair 2		Pair 1	
		Axis 2	Axis 1	Axis 2	Axis 1	Axis 2	Axis 1

The axes are coded in 4 bits and indicate the axis number (1 through 7) according to the order they are programmed.

If the CNC controls the X, Y, Z, B, C axes and G28 BC has been programmed, the AXICOM variable will show the following information:

		Pair 3		Pair 2		Pair 1	
						C	B
0000	0000	0000	0000	0000	0000	0101	0100

TANGAN Variable associated with the tangential control (G45). It indicates the programmed angular position.

Read-write variables

TIMER: This variable allows time, in seconds, indicated by the clock enabled by the PLC to be read or modified. Possible values 0...4294967295

If this variable is accessed, **block preparation is interrupted** and the CNC waits for this command to be executed to resume block preparation.

PARTC: The CNC has a part counter whose count increases every time M30 or M02 is executed and this variable allows it value to be read or modified, which will be given by a number between 0 and 4294967295

If this variable is accessed, **block preparation is interrupted** and the CNC waits for this command to be executed to resume block preparation.

KEY: Returns the code of the last key accepted.

If this variable is accessed, **block preparation is interrupted** and the CNC waits for this command to be executed to resume block preparation.

KEYSRC: This variable allows the origin of keys to be read or modified, possible values being:

0 = Keyboard
1 = PLC
2 = DNC

The CNC only allows modification of this variable if this is at 0.

ANAOn: This variable allows the required analog output (n) to be modified. The value assigned will be expressed in volts and in the ±2.4 format (±10 Volts).

The analog outputs which are free among the eight (1..8) available at the CNC may be modified, the corresponding error being displayed if an attempt is made to write in one occupied.

If this variable is accessed, **block preparation is interrupted** and the CNC waits for this command to be executed to resume block preparation.

13.3 CONSTANTS

Constants are defined as being all those fixed values which cannot be altered by a program. The following are considered as constants:

- Numbers expressed in the decimal system.
- Hexadecimal numbers.
- **PI** (¶) constant.
- Read-only tables and variables as their value cannot be altered with a program.

13.4 OPERATORS

An operator is a symbol which indicates mathematical or logic manipulations which must bemade. The CNC has arithmetic, relational, logic, binary, trigonometric operators and special operators.

Arithmetic operators

+	: add.	P1=3 + 4	—> P1=7
-	: subtraction, also to indicate a negative number	P2=5 - 2 P3=-(2*3)	—> P2=3 —> P3=-6
*	: multiplication	P4=2*3	—> P4=6
/	: division	P5=9/2	—> P5=4.5
MOD	: module (remainder of a division)	P6=7 **MOD** 4	—> P6=3
EXP	: exponential	P7=2 **EXP** 3	—> P7=8

Relational operators

EQ : equal
NE : different
GT : greater than
GE : greater than or equal to
LT : less than
LE : less than or equal to

Logic or binary operators

NOT, OR, AND, XOR: act as logic operators between conditions and as binary operators between variables and constants.

```
IF (FIRST AND GS1 EQ 1) GOTO N100
P5 = (P1 AND (NOT P2 OR P3))
```

Trigonometric functions

SIN	: sine	P1=**SIN** 30	—> P1=0.5
COS	: cosine	P2=**COS** 30	—> P2=0.8660
TAN	: tangent	P3=**TAN** 30	—> P3=0.5773
ASIN	: arc sine	P4=**ASIN** 1	—> P4=90
ACOS	: arc cosine	P5=**ACOS** 1	—> P5=0
ATAN	: arc tangent	P6=**ATAN** 1	—> P6=45
ARG	: ARG (x,y) arc tangent y/x	P7=**ARG**(-1,-2)	—> P7=243.4349

There are two functions for calculating the arc tangent: ATAN which returns the result between ±90° and ARG given between 0 and 360°.

Other functions

ABS	: absolute value	P1=**ABS** -8	—> P1=8
LOG	: decimal logarithm	P2=**LOG** 100	—> P2=2
SQRT	: square root	P3=**SQRT** 16	—> P3=4
ROUND	: rounding up a number	P4=**ROUND** 5.83	—> P4=6
FIX	: integer	P5=**FIX** 5.423	—> P5=5
FUP	: if integer takes integer	P6=**FUP** 7	—> P6=7
	if not, takes entire part + 1	P6=**FUP** 5.423	—> P6=6
BCD	: converts given number to BCD	P7=**BCD** 234	—> P7=564

0010	0011	0100

BIN	: converts given number to binary	P8=**BIN** $AB	—> P8=171

1010	1011

Conversions to binary and BCD are made in 32 bits, it being possible to represent the number 156 in the following formats:

Decimal	156
Hexadecimal	9C
Binary	0000 0000 0000 0000 0000 0000 1001 1100
BCD	0000 0000 0000 0000 0000 0001 0101 0110

13.5 EXPRESSIONS

An expression is any valid combination between operators, constants and variables.

All expressions must be placed between brackets, but if the expression is reduced to an integer, the brackets can be removed.

13.5.1 ARITHMETIC EXPRESSIONS

These are formed by combining functions and arithmetic, binary and trigonometric operators with the constants and variables of the language.

The way to operate with these expressions is established by operator priorities and their associativity:

Priority from highest to lowest Associativity

NOT, functions, - (negative)	from right to left
EXP, MOD	from left to right
*,/	from left to right
+,-(add, subtract)	from left to right
relational operators	from left to right
AND, XOR	from left to right
OR	from left to right

It is advisable to use brackets to clarify the order in which the evaluation of the expression is done.

(P3 = P4/P5 - P6*P7 - P8/P9)
(P3 = (P4/P5)-(P6*P7)-(P8/P9))

The use of repetitive or additional brackets will not produce errors nor will they slow down execution.

In functions, brackets must be used except when these are applied to a numerical constant, in which case they are optional.

(SIN 45) (SIN (45)) both are valid and equivalent.
(SIN 10+5) the same as ((SIN 10)+5).

Expressions can be used also to reference parameters and tables:

(P100 = P9)
(P100 = P(P7))
(P100 = P(P8 + SIN (P8 *20)))
(P100 = ORGX 55)
(P100 = ORGX (12+P9))
(PLCM5008 = PLCM5008 OR 1); selects Single Block execution (M5008=1)
(PLCM5010 = PLCM5010 AND $FFFFFFFE); Frees feedrate Override (M5010=0)

13.5.2 RELATIONAL EXPRESSIONS

These are arithmetic expressions joined by relational operators

(IF **(P8 EQ 12.8)** .... ;Analyzes if the value of P8 is equal to 12.8
(IF **(ABS(SIN(P24)) GT SPEED)** ... ;Analyzes if the sine is greater than the spindle speed.
(IF **(CLOCK LT(P9*10.99))** .... ;Analyzes if the clock count is less than (P9*10.99)

At the same time these conditions can be joined by means of logic operators.

(IF **((P8EQ12.8) OR (ABS(SIN(P24)) GT SPEED)) AND (CLOCK LT (P9*10.99))** ....

The result of these expressions is either true or false.

14. PROGRAM CONTROL STATEMENTS

The control statements available to high-level programming can be grouped as follows:

* Programming statements consisting of:

    Assignment statements
    Display statements
    Enable-disable statements
    Flow control statements
    Subroutine statements
    Statements for generating programs
    Screen customizing statements

* Screen customizing statements

Only one statement can be programmed in each block, and no other additional information may be programmed in this block.

14.1 ASSIGNMENT STATEMENTS

This is the simplest type of statement and can be defined as:

(target=arithmetic expression)

A local or global parameter or a read-write variable may be selected as target. The arithmetic expression may be as complex as required or a simple numerical constant.

(P102 = FZLOY)
(ORGY 55 = (ORGY 54 + P100))

In the specific case of designating a local parameter using its name (A instead of P0, for example) and the arithmetic expression being a numerical constant, the statement can be abbreviated as follows:

(P0=13.7) ==> (A=13.7) ==> (A13.7)

Within a single block, up to 26 assignments can be made to different targets, a single assignment being interpreted as the set of assignments made to the same target.

(P1=P1+P2, P1=P1+P3,P1=P*P4,P1=P1/p5) is the same as (P1=(P1+P2+P3)*P4/P5).

The different assignments which are made in the same block will be separated by commas ",".

14.2 DISPLAY STATEMENTS

(ERROR integer, “error text”

This statement stops the execution of the program and displays the indicated error, it being possible to select this error in the following ways:

(ERROR integer). This will display the error number indicated and the text associated to this number according to the CNC error code (should there be one).

(ERROR integer “error text”). This will display the number and the error text indicated, it being necessary to write the text between quote marks “”.

(ERROR “error text”). This will display the error text only.

The error number may be defined by means of a numerical constant or an arithmetic parameter. When using a local parameter, its numeric format must be used (P0 thru P25 instead of A thru Z).

Programming Examples:

```
(ERROR 5)
(ERROR P100)
(ERROR "Operator error")
(ERROR 3, "Operator error")
(ERROR P120, "Operator error)
```

(MSG “message”)

This statement will display the message indicated between quote marks.

The CNC screen is provided with an area for displaying DNC or user program messages, and always displays the last message received irrespective of where it has come from.

Example:

(MSG “Check tool”)

(DGWZ expression 1, expression 2, expression 3, expression 4, expression 5, expression 6)

The DGWZ instruction (Define Graphic Work Zone) defines the graphics area.

Each expression forming the instruction syntax correspond to one of the limits and they must be defined in millimeters or inches.

expression 1	X minimum
expression 2	X maximum
expression 3	Y minimum
expression 4	Y maximum
expression 5	Z minimum
expression 6	Z maximum

14.3 ENABLING-DISABLING STATEMENTS

(ESBLK and DSBLK)

After executing the mnemonic **ESBLK**, the CNC executes all the blocks which come after as if it were dealing with a single block.

This single block treatment is kept active until it is cancelled by executing the mnemonic **DSBLK**.

In this way, should the program be executed in the SINGLE BLOCK operating mode, the group of blocks which are found between the mnemonics ESBLK and DSBLK will be executed in a continuous cycle, i.e., execution will not be stopped at the end of a block but will continue by executing the following one.

Example:

```
G01 X10 Y10 F800 T1 D1
(ESBLK)                    ; Start of single block
G02 X20 Y20 I20 J-10
G01 X40 Y20
G01 X40 Y40 F10000
G01 X20 Y40 F8000
(DSBLK)                    ; Cancellation of single block
G01 X10 Y10
M30
```

(ESTOP and DSTOP)

After executing the mnemonic **DSTOP**, the CNC enables the Stop key, as well as the Stop signal from the PLC.

It will remain disabled until it is enabled once again by means of the mnemonic **ESTOP**.

(EFHOLD and DFHOLD)

After executing the mnemonic **DFHOLD**, the CNC enables the Feed-Hold input from the PLC.

It will remain disabled until it is enabled once again by means of the mnemonic **EFHOLD**.

14.4 FLOW CONTROL STATEMENTS

The **GOTO** and **RPT** instructions cannot be used in programs that are executed from a PC connected through the serial lines.

(GOTO N(expression))

The mnemonic **GOTO** causes a jump within the same program, to the block defined by the label N(expression).

The execution of the program will continue after the jump, from the indicated block.

The jump label can be addressed by means of a number or by any expression which results in a number.

Example:

```
G00  X0   Y0   Z0  T2 D4
     X10
(GOTO N22)                      ; Jump statement
     X15 Y20                    ; Is not executed
     Y22 Z50                    ; Is not executed
N22  G01  X30  Y40  Z40 F10000  ; Continues execution in this block
     G02 X20  Y40  I-5   J-5
     .....
     .....
```

(RPT N(expression), N(expression))

The mnemonic **RP**T executes, within the same program, the part of the program which exists between the blocks defined by means of the labels **N(expression**).

Both labels can be indicated by means of a number or by any expression which results in a number.

The part of the program selected by means of the two labels must belong to the same program, by first defining the initial block and then the final block.

The execution of the program will continue in the block following the one in which the mnemonic RPT was programmed, once the selected part of the program has been executed.

Example:

```
N10  G00   X10
     Z20
     G01   X5
     G00   Z0
N20        X0
N30  (RPT  N10, N20) N3
N40  G01   X20
     M30
```

When reaching block N30, the program will execute section N10-N20 three times. Once this has been completed, the program will continue execution in block N40.

(IF condition <action1> ELSE <action2>)

This statement analyzes the given condition which must be a relational expression. If the condition is true (result equal to 1), **<action1>** will be executed, otherwise (result equal to 0) **<action2>** will be executed.

Example:

(IF(P8 EQ 12.8) CALL 3 ELSE PCALL 5, A2, B5, D8)

If P8 = 12.8 executes the mnemonic (CALL3)
If P8 <> 12.8 executes the mnemonic (PCALL 5, A2, B5, D8)

The statement can lack the ELSE part, i.e., it will be enough to program IF condition <action1>.

Example:

(IF(P8 EQ 12.8)CALL 3)

Both <action1> and <action2> can be expressions or statements, except for mnemonics IF and SUB.

Due to the fact that in a high level block local parameters can be named by means of letters, expressions of this type can be obtained:

(IF (E EQ 10)M10)

If the condition of parameter P5 (E) having a value of 10 is met, the miscellaneous function M10 will not be executed, since a high level block cannot have ISO code commands. In this case M10 represents the assignment of value 10 to parameter P12, i.e., one can program either:

(IF(E EQ 10)M10) or (IF(P5 EQ 10) P12=10)

14.5 SUBROUTINE STATEMENTS

A subroutine is a part of a program which, being properly identified, can be called from any position of a program to be executed.

A subroutine can be kept in the memory of the CNC as an independent part of a program and be called one or several times, from different positions of a program or different programs.

Only subroutines stored in the CNC's RAM memory can be executed. Therefore, to execute a subroutine stored in the Memkey Card, HD or in a PC connected through the serial lines, it must be copied first into the CNC's RAM memory.

If the subroutine is too large to be copied into RAM, it must be converted into a program and then the EXEC instruction must be used as described in section 14.6

(SUB integer)

The mnemonic **SUB** defines the set of program blocks which are programmed after this block as a subroutine by identifying this subroutine with an integer, between 0 and 9999, which is specified after it:

There can not be two subroutines with the same identification number in the CNC memory, even when they belong to different programs.

(RET)

The mnemonic **RET** indicates that the subroutine which was defined by the mnemonic SUB, finishes in this block.

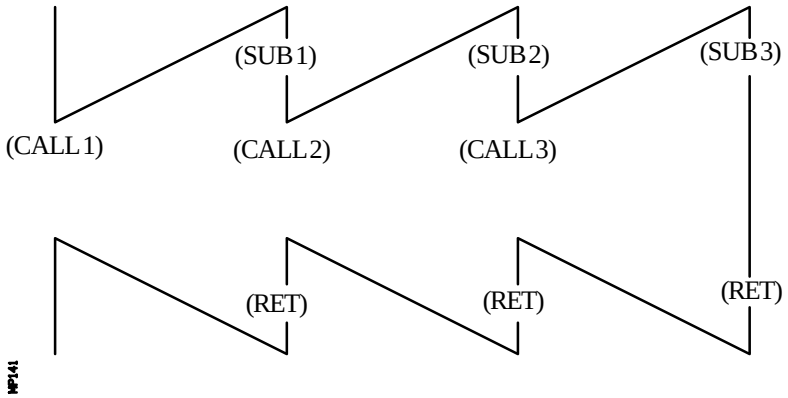
Example:

```
(SUB 12)                ; Definition of subroutine 12
G91  G01  XP0 F5000
     YP1
     X-P0
     Y-P1
(RET)                   ; End of subroutine
```

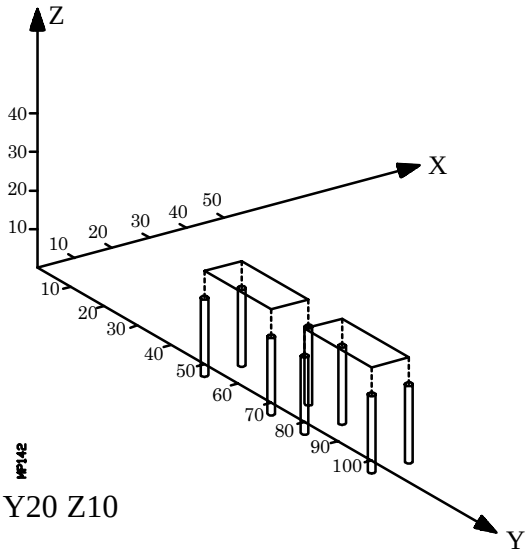
(CALL (expression))

The mnemonic **CALL** makes a call to the subroutine indicated by means of a number or by means of any expression which results in a number.

As a subroutine may be called from a main program, or a subroutine, from this subroutine to a second one, from the second to a third, etc..., the CNC limits these calls to a maximum of 15 nesting levels, it being possible to repeat each of the levels 9999 times.



Example



G90 G00 X30 Y20 Z10
(CALL 10)
G90 G00 X60 Y20 Z10
(CALL 10)
M30

(SUB 10)
G91 G01 X20 F5000
(CALL 11) ; Drilling and threading
G91 G01 Y10
(CALL 11) ; Drilling and threading
G91 G01 X-20
(CALL 11) ; Drilling and threading
G91 G01 Y-10
(CALL 11) ; Drilling and threading
RET

(SUB 11)
G81 G98 G91 Z-8 I-22 F1000 S5000 T1 D1 ; Drilling canned cycle
G84 Z-8 I-22 K15 F500 S2000 T2 D2 ; Threading canned cycle
G80
(RET)

(PCALL (expression), (assignment statement), (assignment statement),...)

The mnemonic **PCALL** calls the subroutine indicated by means of a number or any expression which results in a number. In addition, it allows up to a maximum of 26 local parameters of this subroutine to be initialized.

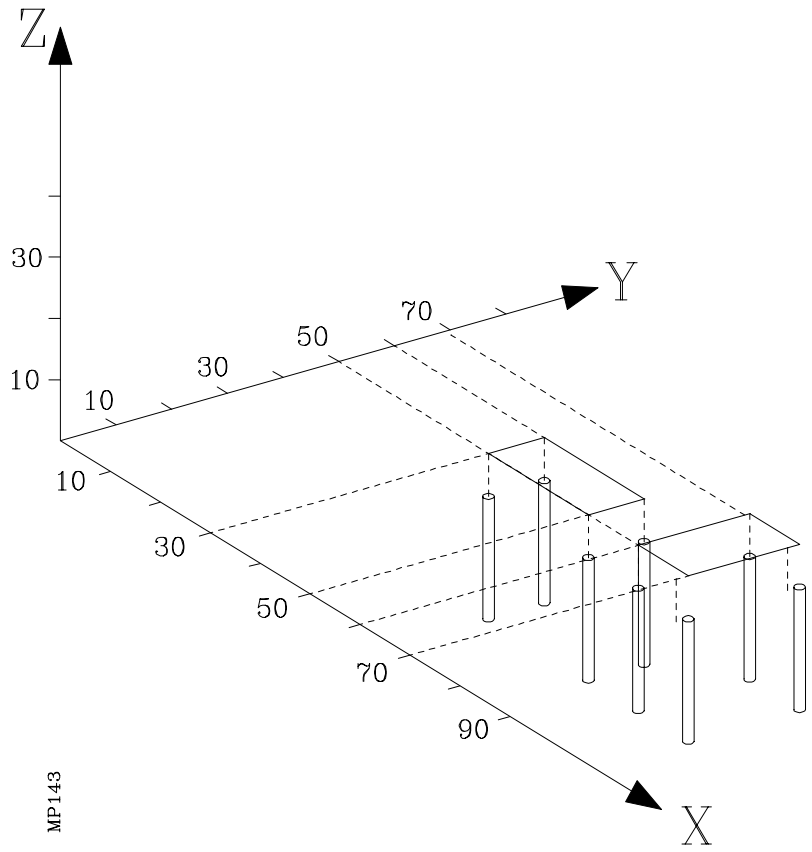
These local parameters are initialized by means of **assignment statements**.

Example: (PCALL 52, A3, B5, C4, P10=20)

In this case, in addition to generating a new subroutine nesting level, a new local parameter nesting level will be generated, there being a maximum of 6 levels of local parameter nesting, within the 15 levels of subroutine nesting.

Both the main program and each subroutine which is found on a parameter nesting level, will have 26 local parameters (P0-P25).

Example:



```
G90 G00 X30 Y50 Z0
(PCALL 10, P0=20, P1=10)                ; or also (PCALL 10, A20, B10)
G90 G00 X60 Y50 Z0
(PCALL 10, P0=10 P1=20)                 ; or also (PCALL 10, A10 B20)
M30

(SUB 10)
G91 G01 XP0 F5000
(CALL 11)
G91 G01 YP1
(CALL 11)
G91 G01 X-P0
(CALL 11)
G91 G01 Y-P1
(CALL 11)
RET

(SUB 11)
G81 G98 G91 Z-8 I-22 F1000 S5000 T1 D1  ; Drilling canned cycle
G84 Z-8 I-22 K15 F500 S2000 T2 D2       ; Threading canned cycle
G80
(RET)
```

(MCALL (expression), (assignment statement), (assignment statement),...)

By means of the mnemonic **MCALL**, any user-defined subroutine (SUB integer) acquires the category of canned cycle.

The execution of this mnemonic is the same as the mnemonic PCALL, but the call is modal, i.e., if another block with axis movement is programmed at the end of this block, after this movement, the subroutine indicated will be executed and with the same call parameters.

If, when a modal subroutine is selected, a movement block with a number of repetitions is executed, for example X10 N3, the CNC will execute the movement only once (X10) and after the modal subroutine, as many times as the number of repetitions indicates.

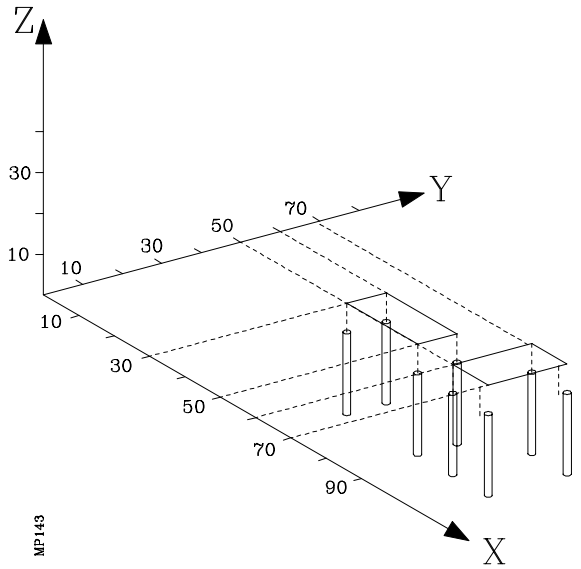
Should block repetitions be chosen, the first execution of the modal subroutine will be made with updated call parameters, but not for the remaining times, which will be executed with the values which these parameters have at that time.

If, when a subroutine is selected as modal, a block containing the MCALL mnemonic is executed, the present subroutine will lose its modal quality and the new subroutine selected will be changed to modal.

(MDOFF)

The mnemonic **MDOFF** indicates that the modal quality acquired by the subroutine with the MCALL mnemonic, finishes in this block.

The use of modal subroutines simplifies programming.



Example:

```
G90 G00 X30 Y50 Z0
(PCALL 10, P0=20, P1=10)
G90 G00 X60 Y50 Z0
(PCALL 10, P0=10 P1=20)
M30

(SUB 10)
G91 G01 XP0 F5000
(MCALL 11)
G91 G01 YP1
G91 G01 X-P0
G91 G01 Y-P1
(MDOFF)
RET)

(SUB 11)
G81 G98 G91 Z-8 I-22 F1000 S5000 T1 D1
G84 Z-8 I-22 K15 F500 S2000 T2 D2
G80
(RET)
```

(PROBE (expression), (assignment statement), (assignment statement),...)

The mnemonic **PROBE** calls the probe cycle indicated by means of a number or any **expression** which results in a number. In addition, it allows the local parameters of this subroutine to be initialized by means of **assignment statements**.

This mnemonic also generates a new level of subroutine nesting.

(DIGIT (expression), (assignment statement), (assignment statement),...)

The mnemonic **DIGIT** calls the digitizing cycle by means of a number or any **expression** which results in a number. It also allows resetting the local parameters of such cycle by means of the **assignment statements**.

The digitized points are sent to the program (in memory or via DNC) previously opened with the following statement:

(OPEN P(expression), (destination directory), A/D, "program comment")

This statement will also generate a new nesting level of subroutines.

(TRACE (expression), (assignment statement), (assignment statement),...)

The mnemonic **TRACE** calls the tracing cycle by means of a number or any **expression** which results in a number. It also allows resetting the local parameters of such cycle by means of the **assignment statements**.

The digitized points are sent to the program (in memory or via DNC) previously opened with the following statement:

(OPEN P(expression), (destination directory), A/D, "program comment")

This statement will also generate a new nesting level of subroutines.

14.5.1 INTERRUPTION SUBROUTINE STATEMENTS

Whenever one of the general interruption logic input is activated, "INT1" (M5024), "INT2" (M5025), "INT3" (M5026) or "INT4 (M5027), the CNC temporarily interrupts the execution of the program in progress and starts executing the interruption subroutine whose number is indicated by the corresponding general parameter.

With INT1 (M5024) the one indicated by machine parameter INT1SUB (P35)
With INT2 (M5025) the one indicated by machine parameter INT2SUB (P36)
With INT3 (M5026) the one indicated by machine parameter INT3SUB (P37)
With INT4 (M5027) the one indicated by machine parameter INT4SUB (P38)

The interruption subroutines are defined like any other subroutine by using the statements: "(SUB integer)" and "(RET)".

The interruption subroutines do not change the level of the local arithmetic parameters; thus they can only contain global arithmetic parameters.

Within an interruption subroutine, it is possible to use the "(REPOS X, Y, Z, ...)" statement described next.

Once the execution of the subroutine is over, the CNC resumes the execution of the program which was interrupted.

(REPOS X, Y, Z, ...)

The REPOS statement must always be used inside an interruption subroutine and facilitates the repositioning of the machine axes to the point of interruption.

When executing this statement, the CNC moves the axes to the point where the program was interrupted.

* The axes are repositioned one at a time.

* It is not necessary to define all the axes, only those to be repositioned.

* The axes forming the main plane move together; thus, it is not required to program both axes since the CNC moves both of them with the first one. The movement is not repeated when defining the second one, it is ignored.

Example: The main plane is formed by the X and Y axes, the Z axis is the longitudinal (perpendicular) axis and the machine uses the C and W axes as auxiliary axes. It is desired to first move the C axis, then the X and Y axes and finally the Z axis..

This repositioning move may be defined in any of the following ways:

(REPOS C, X, Y, Z) **(REPOS C, X, Z)** **(REPOS C, Y, Z)**

If the REPOS statement is detected while executing a subroutine not activated by an interruption input, the CNC will issue the corresponding error message.

14.6 PROGRAM STATEMENTS

With this CNC, from a program in execution one can:

Execute another program ..... Statement (EXEC P.....)
Generate a new program ..... Statement (OPEN P.....)
Add blocks to an existing program ..... Statement (WRITE.....)

(EXEC P(expression), (directory)

The **EXEC P** statement executes the part-program of the indicated directory

The part-program may be defined by a number or any **expression** resulting in a number.

By default, the CNC assumes that the part-program is in the CNC's RAM memory. If it is in another device, it must be indicated in (**directory**).

CARDA	in the "Memkey CARD"
HD	on the hard disk
DNC1	at a PC connected through serial line 1
DNC2	at a PC connected through serial line 2

(OPEN P(expression), (destination directory), A/D, "program comment")

This statement starts editing a part-program whose number will be given by any number or **expression** resulting in a number.

By default, the new part-program edited will be stored in the CNC's RAM memory. To store it another device, it must be indicated in (**destination directory**).

CARDA	in the "Memkey CARD"
HD	on the hard disk
DNC1	at a PC connected through serial line 1
DNC2	at a PC connected through serial line 2

Parameter **A/D** is used when the program to be edited already exists.

A	The CNC appends the new blocks after the ones already existing
D	The CNC deletes the existing program and starts editing a new one.

A **program comment** may also be associated with it. This comment will later be displayed next to it on the program directory.

The OPEN statement is very useful when digitizing parts because it allows generating a program from a program already in execution. That generated program will depend on the values assumed by the program being executed.

To edit blocks, the WRITE statement must be used as described next.

Notes: If the program to be edited already exists and the A/D parameters are not defined, the CNC will display an error message when executing the block.

The program opened with the OPEN statement is closed when executing an M30, or another OPEN statement and after an Emergency or Reset.

From a PC, only programs stored in the CNC'S RAM memory, in the CARD A, or in the Hard Disk module can be opened

(WRITE <block text>)

The mnemonic **WRITE** adds, after the last block of the program which began to be edited by means of the mnemonic OPEN P, the information contained in **<block text>** as a new program block.

If parametric programming is used within the <block text> and it has been edited in ISO code, when executing the block, all the parameters (global and local) are replaced with the value they have at time. Blocks edited in high level language are NOT replaced.

In the customizing programs edited by the operator, the IB instruction may be used so the blocks edited in high level assume the parameter value.

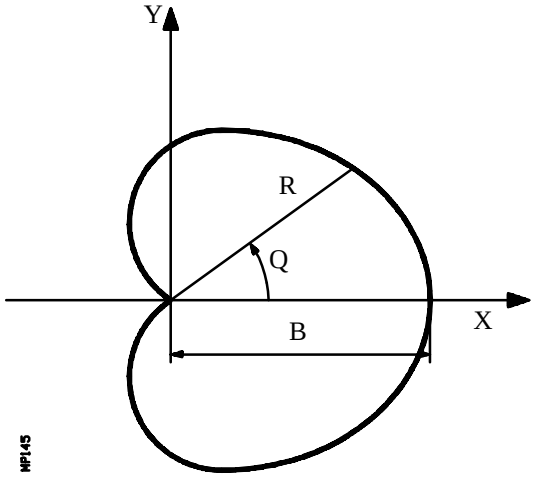
Examples for P100=10, P101=20 y P102=55

```
(WRITE G1 XP100 YP101 F100)            =>  G1 X10 Y20 F100
(WRITE (IF (P100 EQ P101) CALL 3))     =>  (IF (P100 EQ P101) CALL 3)
(WRITE (SUB P102))                     =>  (SUB P102)
(IB1=(P102))
(WRITE (SUB (IB1))                     =>  (SUB 55)
```

If the mnemonic WRITE is programmed without having programmed the mnemonic OPEN previously, the CNC will display the corresponding error, except when editing a user customized program, in which case a new block is added to the program being edited.

Example of the creation of a program which contains several points of a cardioid whose formula is:

$$R=B \cos (Q/2)$$



Subroutine number 2 is used, its parameters having the following meaning:

A or P0	Value of angle Q.
B or P1	Value of B
C or P2	Angular increment for calculation
D or P3	Axis feedrate

A way to use this example could be:

```
G00 X0 Y0
G93
(PCALL 2, A0, B30, C5, D500)
M30
```

Program generation subroutine:

```
(SUB 2)
(OPEN P12345)                ; Starts editing of program P12345
```

14.7 SCREEN CUSTOMIZING STATEMENTS (GRAPHIC EDITOR)

Customizing statements may be used only when customizing programs made by the user.

These customizing programs must be stored in the CNC'S RAM memory and they may utilize the "Programming Statements" and they will be executed in the special channel designed for this use. The program selected in each case will be indicated in the following general machine parameters.

In “USERDPLY” the program to be executed in the Execution Mode will be indicated.

In “USEREDIT” the program to be executed in the Editing Mode will be indicated.

In “USERMAN” the program to be executed in the Manual (JOG) Mode will be indicated.

In “USERDIAG” the program to be executed in the Diagnosis Mode will be indicated.

The customizing programs may have up to five nesting levels besides their current one. Also, the customizing statements do not allow local parameters, nevertheless all global parameters may be used to define them.

(PAGE (expression))

The mnemonic **PAG**E displays the page number indicated by means of a number or by means of any **expressio**n which results in a number.

User-defined pages will be from page 0 to page 255 and will be defined from the CNC keyboard in the Grahic Editor mode and as indicated in the Operating Manual.

System pages will be defined by a number greater than 1000. See the corresponding appendix.

(SYMBOL (expression 1), (expression 2), (expression 3))

The mnemonic **SYMBO**L displays the symbol whose number is indicated by means of the value of **expression 1** once this has been evaluated.

Its position on screen is also defined by **expression 2** (column) and by **expression 3** (row).

Expression 1, expression 2 and expression 3 may contain a number or any expression which results in a number.

The CNC allows to display any user-defined symbol (0-255) defined at the CNC keyboard in the Graphic Editor mode such as is indicated in the Operating Manual.

In order to position it within the display area its pixels must be defined, 0-639 for columns (expression 2) and 0-335 for rows (expression 3).

(IB (expression) = INPUT “text”, format))

The CNC has 26 data entry variables (IBO-1B25)

The **I**B mnemonic displays the text indicated in the data input window and stores the data input by the user in the entry variable indicated by means of a number or by means of any **expression which results in a number**.

The wait for data entry will only occur when programming the format of the requested data. This **format** may have a sign, integer part and decimal part.

If it bears the "minus" sign, it will allow positive and negative values, and if it does not have a sign, it will only allow positive values.

The integer part indicates the maximum number of digits (0-6) desired to the left of the decimal point.

The decimal part indicates the maximum number of digits (0-5) desired to the right of the decimal point.

If the numerical format is not programmed; for example: (IB1 =INPUT "text"), the mnemonic will only display the indicated text without waiting for the data to be entered.

(ODW (expression 1), (expression 2), (expression 3))

The mnemonic **ODW** defines and draws a white window on the screen with fixed dimensions (1 row and 14 columns).

Each mnemonic has an associated number which is indicated by the value of **expression 1** once this has been evaluated.

Likewise, its position on screen is defined by **expression 2** (row) and by **expression 3** (column).

Expression 1, expression 2 and expression 3 may contain a number or any expression which results in a number.

The CNC allows 26 windows (0-25) to be defined and their positioning within the display area, providing 21 rows (0-20) and 80 columns (0-79).

(DW(expression 1) = (expression 2), DW (expression 3) = (expression 4),...)

The mnemonic **DW** displays in the window indicated by the value for expression 1, expression 3, .. once they have been evaluated, the numerical data indicated by expression 2, expression 4, ...

Expression 1, expression 2, expression 3, ... may contain a number or any expression which may result in a number.

The following example shows a dynamic variable display:

```
      (ODW 1,6,33)                ; Defines data window 1
      (ODW 2,14,33)               ; Defines data window 2
N10   (DW1=DATE,DW2=TIME)         ; Displays the date in window 1 and the time in 2
      (GOTO N10)
```

The CNC allows displaying the data in decimal, hexadecimal and binary format. The following instructions are available:

(DW1 = 100)	Decimal format. Value "100" displayed in window 1.
(DWH2=100)	Hexadecimal format. Value "64" displayed in window 2.
(DWB3=100)	Binary format. Value "01100100" displayed in window 3.

When using the binary format, the display is limited to 8 digits in such a way that a value of "11111111" will be displayed for values greater than 255 and the value of "10000000" for values more negative than -127.

Besides, the CNC allows the number stored in one of the 26 data input variables (IB0-IB25) to be displayed in the requested window.

The following example shows a request and later display of axis feedrate.

```
(ODW3,4,60)                 ; Defines data window 3
(IB1=INPUT"Axis feed:",5.4) ; Axis feedrate request
(DW3=IB1)                   ; Displays feedrate in window 3
```

(SK(expression 1) = “text1” (expression 2) = “text 2”, ...)

The mnemonic **SK** defines and displays the new softkey menu indicated.

Each of the **expressions** will indicate the softkey number which it is required to modify (1-7, starting from the left) and the **texts** which it is required to write in them.

Expression 1, expression 2, expression 3, ... may contain a number or any expression which may result in a number.

Each text will allow a maximum of 20 characters which will be shown on two lines of 10 characters each. If the text selected has less than 10 characters, the CNC will center it on the top line, but if it has more than 10 characters the programmer will center it.

Examples:

(SK 1=”HELP”, SK 2=”MAXIMUM COORDINATE”)

(SK 1=”FEEDRATE”,SK 2=”_MAXIMUM__COORDINATE”)

Warning:

If while a standard CNC softkey menu is active, one or more softkeys are selected via high level language statement: "**SK**", the CNC will clear all existing softkeys and it will only show the selected ones.

If while a user softkey menu is active, one or more softkeys are selected via high level language statement "**SK**", the CNC will only replace the selected softkeys leaving the others intact.

(WKEY)

The mnemonic **WKEY** stops execution of the program until the key is pressed.

The pressed key will be recorded in the KEY variable.

Example

```
....
....
(WKEY)                          ; Wait for key
8IF KEY EQ $FC00 GOTO N1000     ; If key F1 has been pressed, continue in
                                  N1000
....
....
```

(WBUF “text”, (expression))

The **WBUF** statement can only be used when editing a program in the user channel.

This mnemonic may be programmed in two ways:

(WBUF “text”, (expression))

This statement adds the text and value of the expression once this has been evaluated, to the block which is being edited and within the data input window.

(Expression) may contain a number or any expression which results in a number.

It will be optional to program the expression, but it will be required to define the text. If no text is required, “” must be programmed.

Examples for P100=10 (WBUF "X", P100) => X10
(WBUF "XP100") => XP100

(WBUF)

Enters into memory, adding to the program being edited and after the cursor position, the block being edited by means of (WBUF "text", (expression)). It also clears the editing buffer in order to edit a new block.

This allows the user to edit a complete program without having to quit the user editing mode after each block and press ENTER to "enter" it into memory.

Example:

```
(WBUF”(PCALL 25,”)          ; Adds “(PCALL 25,” to the block being
                              edited
(IB1=INPUT “Parameter A:”,-5.4)  ; Request of Parameter A
(WBUF “A=”,IB1)             ; Adds “A=(value entered) to the block being
                              edited.
(IB2=INPUT”Parameter B:”,-5.4)   ; Request of Parameter B
(WBUF”,B=”,IB2              ; Adds “B=(value entered)” to the block being
                              edited
(WBUF”)”)                   ; Adds “)” to the block being edited
(WBUF)                      ; Enters the edited block into memory
------------
------------
```

After executing this program the block being edited contains:

(PCALL 25, A=23.5, B=-2.25)

(SYSTEM)

The mnemonic **SYSTEM** stops execution of the user customized program and returns to the corresponding standard menu of the CNC.

Customizing program example

The following customizing program must be selected as user program associated to the Editing Mode.

After selecting the Editing Mode and pressing the USER softkey, this program starts executing and it allows assisted editing of 2 user cycles. This editing process is carried out a cycle at a time and as often as desired.

```
; Displays the initial editing page (screen)
N0   (PAGE 10)

; Sets the softkeys to access the various modes and requests a choice
     (SK 1="CYCLE 1",SK 2="CYCLE 2",SK 7="EXIT")
N5   (WKEY)                                        ;Request a key
     (IF KEY EQ $FC00 GOTO N10)                    ;Cycle 1
     (IF KEY EQ $FC01 GOTO N20)                    ;Cycle 2
     (IF KEY EQ $FC06 SYSTEM ELSE GOTO N5)         ;Quit or request a key

; CYCLE 1
; Displays page 11 and defines 2 data entry windows
N10  (PAGE 11)
     (ODW 1,10,60)
     (ODW 2,15,60)

;Editing
     (WBUF "(PCALL 1,")      ; Adds (PCALL 1,  to the block being edited

     (IB 1=INPUT "X:",-6.5)  ; Requests the value of X
     (DW 1=IB1)              ; Data window 1 shows the entered value
     (WBUF "X",IB1)          ; Adds X (entered value) to the block being edited

     (WBUF ",")              ; Adds ,  to the block being edited

     (IB 2=INPUT "Y:",-6.5)  ; Requests the value of Y
     (DW 2=IB2)              ; Data window 2 shows the entered value
     (WBUF "Y",IB2)          ; Adds Y (entered value) to the block being edited

     (WBUF ")")              ; Adds )  to the block being edited

     (WBUF)                  ; Enters the edited block into memory. For example: (PCALL 1, X2, Y3)

     (GOTO N0)
```

;(This sample program continues on next page)

*; **CYCLE 2***

; Displays page 12 and defines 3 data entry windows

```
N20  (PAGE 12)
     (ODW 1,10,60)
     (ODW 2,13,60)
     (ODW 3,16,60)

;Editing
     (WBUF "( PCALL 2,")      ; Adds (PCALL 2, to the block being edited

     (IB 1=INPUT "A:",-6.5)   ; Requests the value of A
     (DW 1=IB1)               ; Data window 1 shows the entered value
     (WBUF "A",IB1)           ; Adds A (entered value) to the block being edited

     (WBUF ",")               ; Adds ,  to the block being edited

     (IB 2=INPUT "B:",-6.5)   ; Requests the value of B
     (DW 2=IB2)               ; Data window 2 shows the entered value
     (WBUF "B",IB2)           ; Adds B (entered value) to the block being edited

     (WBUF ",")               ; Adds ,  to the block being edited

     (IB 3=INPUT "C:",-6.5)   ; Requests the value of C
     (DW 3=IB3)               ; Data window 3 shows the entered value
     (WBUF "C",IB3)           ; Adds C (entered value) to the block being edited

     (WBUF ")")               ; Adds )  to the block being edited

     (WBUF )                  ;Enters the edited block into memory. Example: (PCALL 2, A3, B1, C3)

     (GOTO N0)
```

15. *DIGITIZING CYCLES*

This CNC offers the following digitizing cycles:

1 Digitizing cycle in a grid pattern.
2 Digitizing cycle in an arc pattern.

These cycle must be programmed by means of the High Level Language instruction **DIGIT** and its programming format is:

(DIGIT (expression), (assignment statement), ...)

This statement calls upon the indicated digitizing cycle by means of a number or an expression resulting in a number. It also allows presetting its parameters with the desired values by using assignment statements.

General considerations

All movements of these digitizing cycles must be made along the X, Y, or Z axes and the work plane must be formed by two of these axes (XY, XZ, YZ, YX, ZX, ZY). The other axis must be perpendicular to this plane and it must be selected as longitudinal axis.

The machining conditions for the digitizing cycle must be defined before calling it.

During the execution of a digitizing cycle, the coordinates of the collected (probed) points are stored in a program. This program must be "opened" before calling the cycle by means of the (OPEN P) statement.

If instead of storing the digitized data in the program memory of the CNC it is desired to send it out to a peripheral or computer via DNC, it must be indicated so when defining the (OPEN P) statement.

It is advisable to indicate the machining conditions of the digitized program (opened with the (OPEN P) statement) by using the (WRITE) statement on the necessary blocks of the digitizing cycle.

Once the digitizing cycle has finished, the probe will be positioned where it was before executing the cycle.

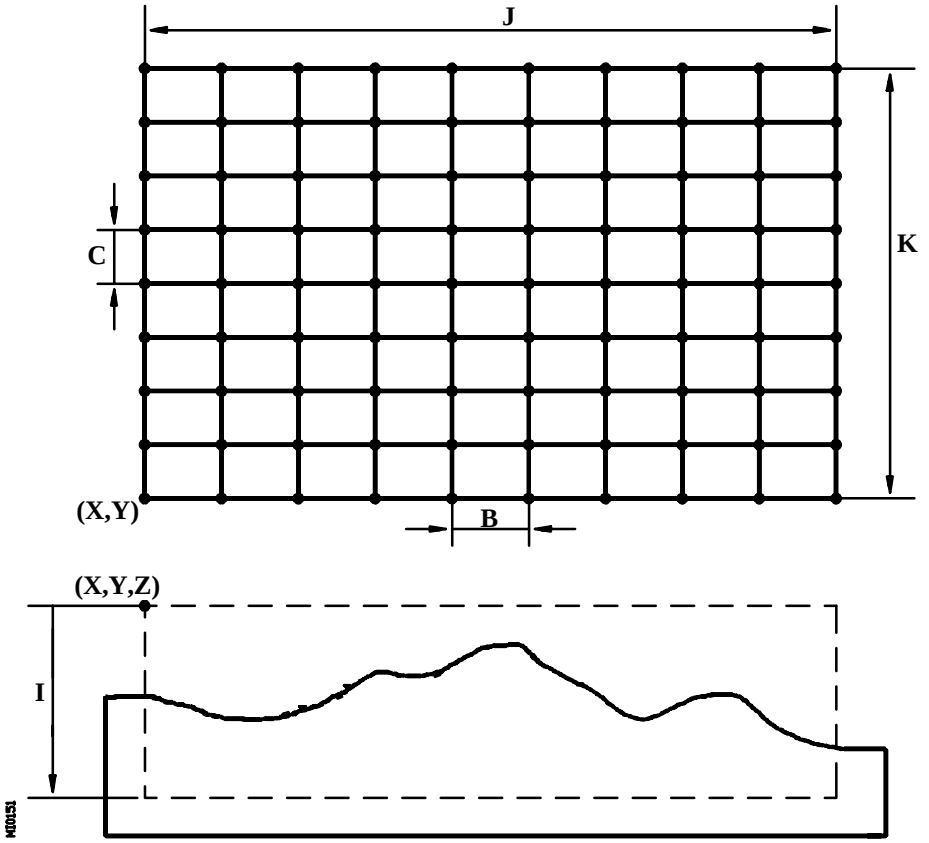
The execution of a digitizing cycle does not alter the history of the previous "G" functions.

It must be borne in mind that the program blocks generated by the digitizing cycle are all positioning blocks. Therefore, to end the generated program, another block containing an M02 or M30 must be added.

15.1 DIGITIZING CYCLE IN A GRID PATTERN

The programming format is as follows:

(DIGIT 1, X, Y, Z, I, J, K, B, C, D, F)



X±5.5 Theoretical position value, along the abscissa axis, of the first digitized point. It must be defined in absolute coordinates and it must coincide with one of the corners of the grid.

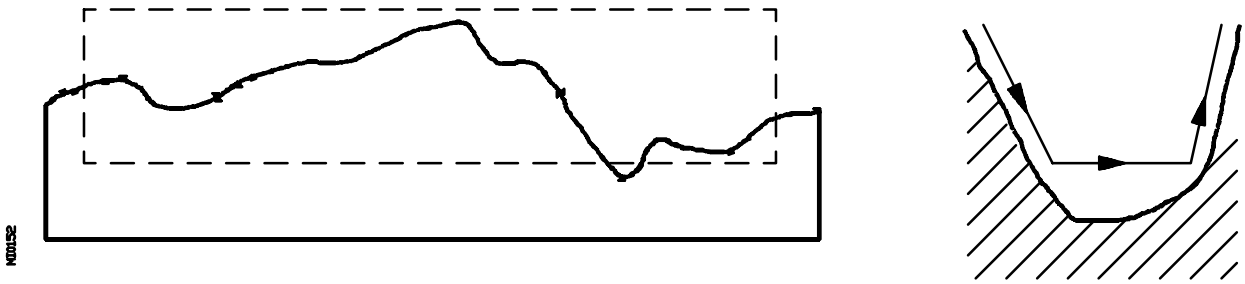
Y±5.5 Theoretical position value, along the ordinate axis, of the first digitized point. It must be defined in absolute coordinates and it must coincide with one of the corners of the grid.

Z±5.5 Theoretical position value, along the probing axis, where the probe will be positioned before starting to digitize. It must be defined in absolute coordinates.

When defining this position value, both the maximum height of the part and the clearance to be maintained with respect to it must be taken into account.

I±5.5 Defines the maximum probing depth and it is referred to the position value assigned to parameter Z.

If a portion of the part is outside this zone, the cycle will not collect the values of its points but it will continue with the digitizing cycle without issuing an error message.



If a 0 value is assigned to this parameter, the CNC will display the corresponding error message.

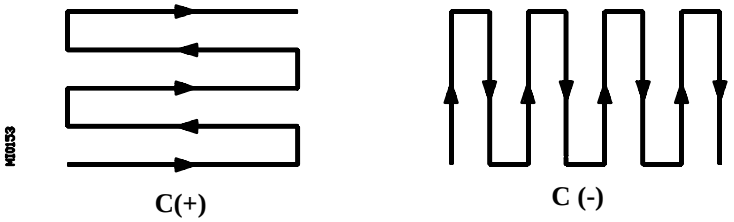
J±5.5 Defines the length of the grid along the abscissa axis. The positive sign indicates that the grid is located to the right of the (X,Y) point and the negative sing indicates that the grid is located to the left of that point.

K±5.5 Defines the length of the grid along the ordinate axis. The positive sign indicates that the grid is located above the (X,Y) point and the negative sing indicates that the grid is located below that point.

B 5.5 Defines the digitizing step along the abscissa axis. It must be programmed with a positive value greater than 0.

C±5.5 Defines the digitizing step along the ordinate axis.

If programmed with a positive value, the digitizing of the grid is carried out following the abscissa axis and if negative, following the ordinate axis.

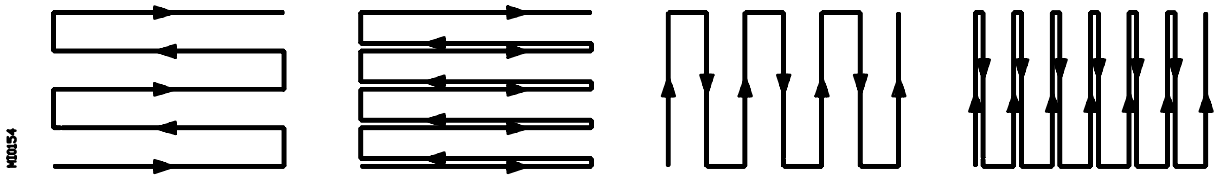


If a value of 0 is programmed, the CNC will show the corresponding error message.

D Indicates how the grid will be "swept" according to the following code:

0 = It will be digitized in both directions (zig-zag).
1 = It will be digitized only in one direction.

If not programmed, the cycle will assume a value of D=0.



F5.5 Defines the probing feedrate in mm/min or inches min.

Basic operation

1.- The probe is positioned at the point defined by parameters X,Y and Z.

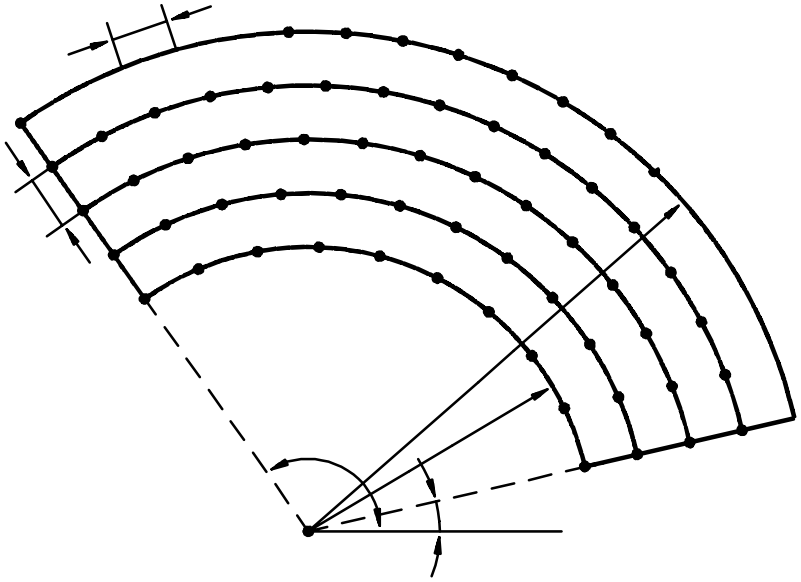
2.- The probe moves along the probing axis until touching the part.

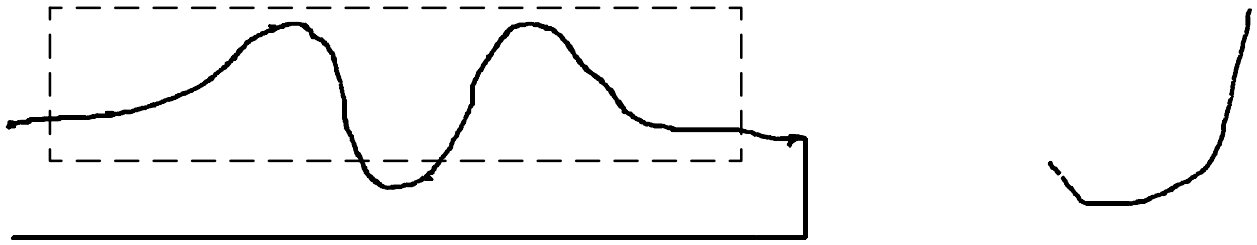
3.- The CNC will generate a new block in the program previously opened with the (OPEN P) statement. This block will indicate the position values of the X, Y and Z axes at this point.

4.- The probe will "follow" the part along the programmed path generating a new block every time the probe touches the part.

5.- Once the canned cycle is finished, the probe will return to the cycle calling point. This move consists of:

* Movement of the probe along the probing axis.
* Movement of the probe in the main work plane.

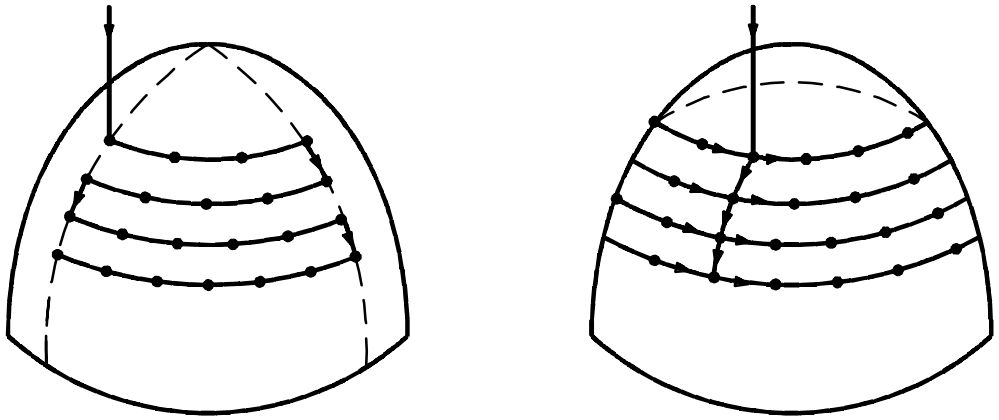




B 5.5 Defines the angular position of the other end of the arc zone (sector) with respect to the abscissa axis.

When defining parameters A and B, it must be borne in mind that the initial digitizing path is followed in the counter-clockwise direction.

When not programming a complete circle, the digitizing paths will be followed in both directions (in a zig-zag manner) and, when programming a complete circle, they will be scanned concentrically.



If not programmed, a value of B=360 will be assumed.

To digitize a complete circle, A and B must be assigned the same value or none at all so the default values are assumed (A0 B360).

C 5.5 Defines the digitizing step. That is, the distance between consecutive arcs and between consecutive points.

F5.5 Defines the probing feedrate in mm/min or inches min.

Basic operation

1.- The probe is positioned at the point defined by parameters X,Y and Z.

2.- The probe moves along the probing axis until touching the part.

3.- The CNC will generate a new block in the program previously opened with the (OPEN P) statement. This block will indicate the position values of the X, Y and Z axes at this point.

4.- The probe will "follow" the part along the programmed path generating a new block every time the probe touches the part.

5.- Once the canned cycle is finished, the probe will return to the cycle calling point. This move consists of:

* Movement of the probe along the probing axis.
* Movement of the probe in the main work plane.

16. TRACING AND DIGITIZING

16.1 INTRODUCTION

In order to clarify the terminology used throughout this chapter, the following definitions are provided:

Tracing: The probe moves following the indicated path and keeping its stylus in contact with the model surface at all times.

Copying: It requires a machine with a second spindle or with a copying arm where the tracing probe is mounted while the machining tool goes on the main spindle.

Copying consists in machining a part while tracing a model. The machined part will be a copy of the traced model.

Digitizing: Consists in capturing the machine coordinates of the part while being traced and send them to a file previously opened by means of the "OPEN P" instruction.

In order to digitize, the tracing function G23 must be activated whether the part is going to be copied or not.

The model can be traced and digitized in two ways:

Manually.

It allows the operator to move the probe by hand on and along the surface of the model.

Automatically.

The probe movements are controlled by the CNC which offers the following choices:

By activating one of these canned cycles:

TRACE 1	Tracing/Digitizing in a grid pattern.
TRACE 2	Tracing/Digitizing in an arc pattern.
TRACE 3	Profile Tracing/Digitizing in the plane.
TRACE 4	3-D Profile Tracing/Digitizing (in space)
TRACE 5	Profile Tracing/Digitizing with polygonal sweep

By activating the tracing (G23) and digitizing (G24) functions.

In this case the path to be followed by the probe must be defined. The available options are:

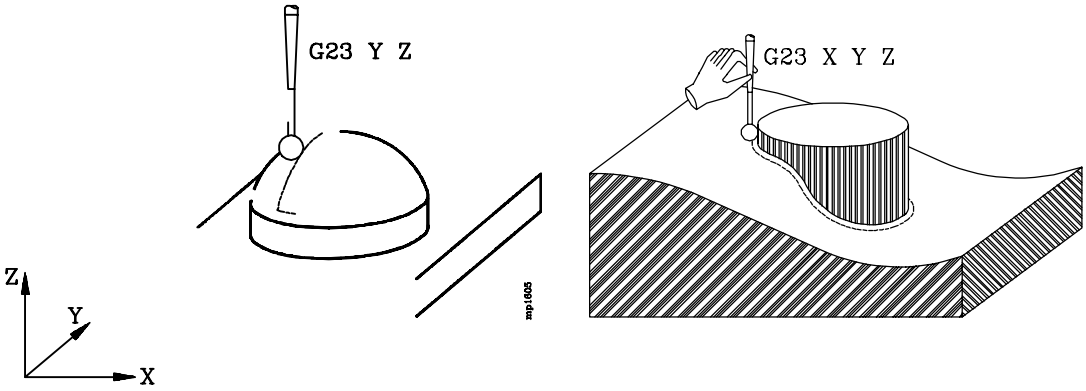
One-dimensional tracing/digitizing.
Two-dimensional tracing/digitizing.
Three-dimensional tracing/digitizing.

All these tracing/digitizing types are being described next.

*** Manual Tracing / Digitizing.**

It allows the operator to move the probe by hand on and long the surface of the model being possible to limit the manual movement of the probe to 1, 2 or 3 axes.

With this type of tracing it is possible to capture points of the model, to make parallel tracing passes, two-dimensional or three-dimensional contouring, roughing operations, etc.



With this option it is possible to digitize the model either point by point or continuously.

The continuous digitizing is carried out by the CNC according to the values assigned to the digitizing parameters. Function G24.

To digitize point by point, function G24 must be defined without parameters. The point capture is carried out by the operator by pressing the "READ-POINT-BY-POINT" softkey or by activating an external push-button.INTRODUCTION

*** One-dimensional Tracing / Digitizing.**

Is the most common type of tracing.

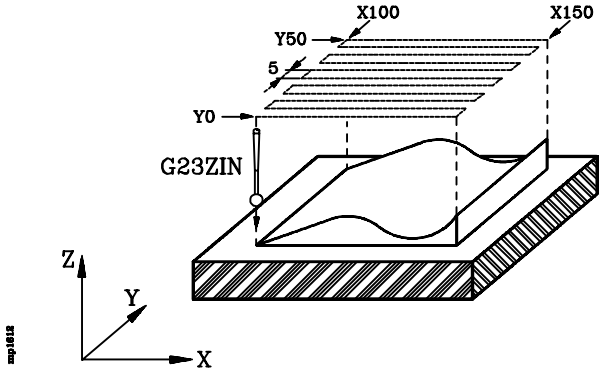
When defining function G23, it must be indicated which axis, controlled by the CNC, sweeps the model.

The path to be followed by the tracing probe will be established by the other two axes by either programming it in ISO code or by jogging those axes using the JOG keys or the electronic handwheel.

This option permits digitizing the model continuously (as opposed to point-by-point). It will be controlled by the CNC depending on the values assigned to digitizing parameters. Function G24.

Example:

The tracing zone is delimited between (X100 Y0) and (X150 Y50), the Z axis being the probing axis.



G90 G01 X100 Y0 Z80 F1000	
(OPEN P234)	Program receiving the data
(WRITE G90 G01 G05 F1000)	
G23 Z I-10 N1.2	Tracing **ON**
G24 L8 E5 K1	Digitizing **ON**
N10 G91 X50	Define the sweeping path (pattern)
Y5	"
X-50	"
N20 Y5	"
(RPT N10,N20) N4	"
X50	"
G25	Tracing and digitizing **OFF**
M30	

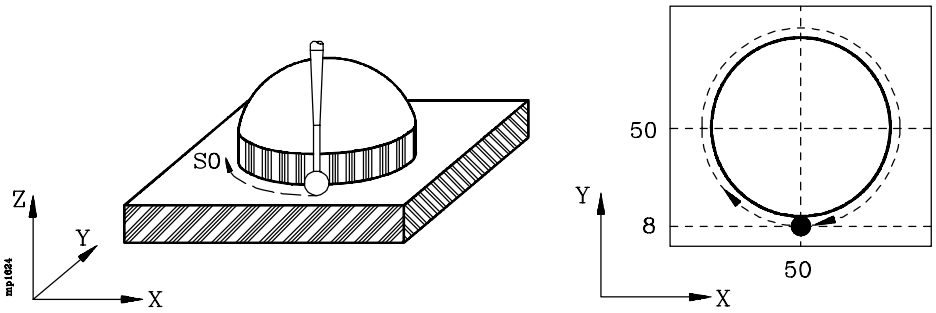
*** Two-dimensional Tracing / Digitizing.**

It contours the model. To do this, it is necessary to define the 2 axes which, being controlled by the CNC, follow the profile.

The contour, defined by function G27, may be either closed (where the initial and final points are the same) or open (where the initial and final points are not the same).

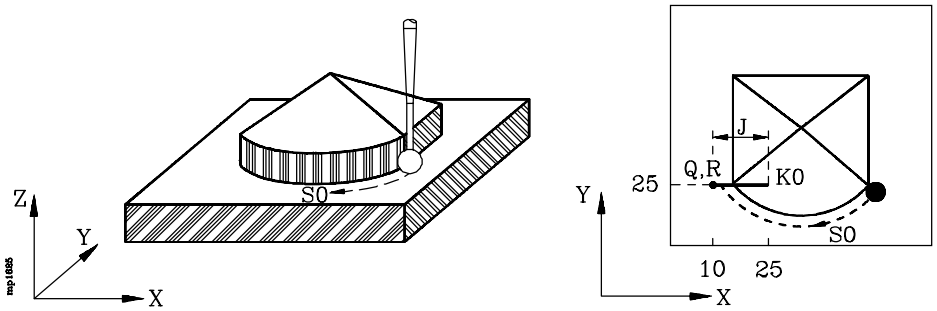
With this option it is possible to carry out a continuous digitizing of the model which will be controlled by the CNC depending on the values assigned to the digitizing parameters. Function G24

Example of a closed contour:




```
G23 XY I50 J8 N0.8        ;Two-dimensional tracing definition
G24 L8 E5 K1              ;Digitizing definition
G27 S0                    ;Closed contour definition
G25                       ;Deactivate tracing and digitizing
```

Example of an open contour:




```
G23 XY I60 J20 N0.8       ;Two-dimensional tracing definition
G24 L8 E5 K1              ;Digitizing definition
G27 S0 Q10 R25 J15 K0     ;Open contour definition
G25                       ;Deactivate tracing and digitizing
```

*** Three-dimensional Tracing / Digitizing.**

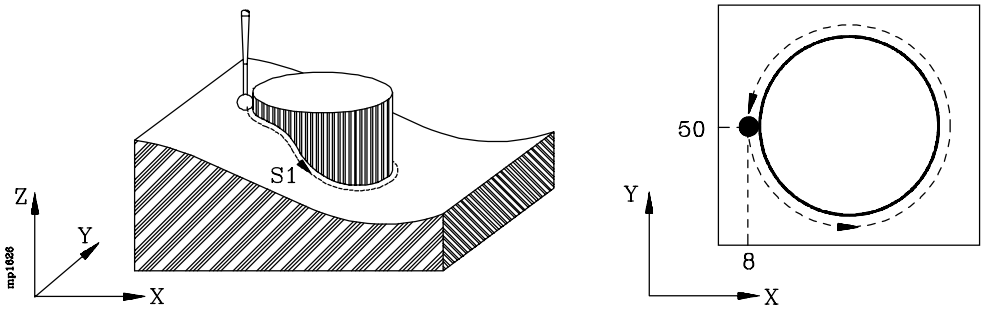
The profile contouring is carried out by three axes which are controlled by the CNC.

There must always be a surface for the probe to touch. The maximum slope of this sweeping surface depends on the sweeping feedrate and the nominal deflections. The greater the sweeping feedrate the flatter the surface must be.

The contour, defined by function G27, may be either closed (where the initial and final points are the same) or open (where the initial and final points are not the same).

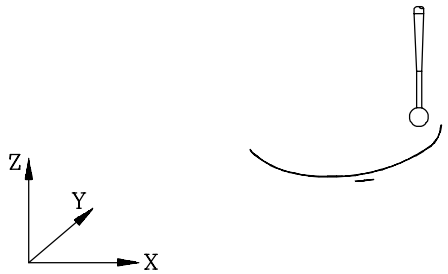
With this option it is possible to carry out a continuous digitizing of the model which will be controlled by the CNC depending on the values assigned to the digitizing parameters. Function G24

Example of a closed contour:




```
G23 XYZ I8 J50 K75 N0.8 M0.5     ;Three-dimensional tracing definition
G24 L8 E5 K1                     ;Digitizing definition
G27 S1                           ;Closed contour definition
G25                              ;Deactivate tracing and digitizing
```

Example of an open contour:




```
G23 XYZ I20 J50 K45 N0.8 M0.5    ;Three-dimensional tracing definition
G24 L8 E5 K1                     ;Digitizing definition
G27 S1 Q80 R40 J25 K0            ;Open contour definition
G25                              ;Deactivate tracing and digitizing
```

*** Tracing / Digitizing canned cycles**

The tracing/digitizing canned cycles offered by this CNC are based on the types of tracing described earlier and they are the following:

TRACE 1 Tracing / digitizing in a grid pattern
TRACE 2 Tracing / digitizing in an arc pattern.
TRACE 3 Profile tracing / digitizing in the plane
TRACE 4 3-D Profile tracing / digitizing (in space)
TRACE 5 Tracing / digitizing with polygonal sweep.

They are programmed by means of the high level instruction **TRACE.** The cycle number may be indicated either by a number (1, 2, 3, 4, 5) or by an expression whose result is one of these numbers.

They all have a series of parameters defining the tracing path and the digitizing conditions.

To just trace the part without digitizing it, the digitizing parameters must be set to "0".

To digitize the model, besides setting the digitizing parameters, it is required to open the program storing the digitized data by means of the "**OPEN P**" statement.

16.1.1 GENERAL CONSIDERATIONS

The CNC offers the following preparatory functions to trace / digitize parts:

G26 Calibrate the tracing probe
G23 Activate the tracing function
G24 Activate the digitizing function
G27 Define the tracing contour
G25 Deactivate the tracing / digitizing function

It also offers the following tracing canned cycles:

TRACE 1 Tracing / digitizing in a grid pattern
TRACE 2 Tracing / digitizing in an arc pattern.
TRACE 3 Profile tracing / digitizing in the plane
TRACE 4 3-D Profile tracing / digitizing (in space)
TRACE 5 Tracing / digitizing with polygonal sweep.

About tracing

While tracing the model, the CNC only controls the movements of the X, Y and Z axes. Thus, the main plane (work plane) must be formed by two of these axes (XY, XZ, YZ, YX, ZX, ZY). The other axis must be perpendicular to that plane and set as longitudinal axis.

The tracing probe must always be mounted on that perpendicular axis.

The tracing probe must be calibrated (G26) every time it is installed on the machine, it is changed or reoriented and every time the CNC is powered-up

Once function G23 is executed (tracing), the CNC maintains the probe in contact with the surface of the model following the selected path at all times.

When tracing automatically (not by hand), it is necessary to define the path to be followed by the tracing probe by either programming it in ISO code or by jogging the axes with the JOG keys or with the electronic handwheel.

To deactivate the tracing previously activated with function G23, execute function G25.

When executing one of the tracing / digitizing canned cycles it is not necessary to execute function G23, G25, or to define the tracing path since it is already taken care of by the canned cycle itself.

When copying (machining while tracing) it is not possible to compensate for probe deflection. Therefore, it is recommended to use a machining tool whose radius is equal to or smaller than the radius of the probe tip (ball) minus the amount of stylus deflection being applied.

For example: When using a 10 mm diameter ball (5 mm radius) with a maximum deflection of 1mm, a 8mm-diameter (4 mm radius) tool should be used

About digitizing

Digitizing consists in taking (capturing) points (coordinates) of the machine during the tracing process and send them to a file previously opened with the "**OPEN P**" statement.

In order to digitize a model it is necessary to either execute one of the tracing/digitizing canned cycles (TRACE) or, define the path to be followed by the probe on the surface of the model once the tracing (G23) and digitizing (G24) functions have been activated.

The CNC captures points on the model surface depending on the parameters indicated when defining function G24 or, in the JOG mode, whenever the operator presses the external push-button or corresponding softkey.

During the digitizing of the model, the CNC only controls the movements of the X, Y and Z axes. Therefore, the generated program blocks will only contain the information on some or all three axes: X, Y and Z.

Besides, the CNC takes into account the deflections of the probe when calculating the coordinates of the new digitized point.

The CNC does not take any points automatically while the probe is searching for the model or when it is off its surface.

16.2 G26. CALIBRATION OF THE TRACING PROBE

This function executes an internal calibration cycle which permits compensating for the possible lack of parallelism between the probe axes and those of the machine.

It is recommended to perform this calibration every time the probe is installed on the machine, it is changed or reoriented and every time the CNC is powered up.

In order to calibrate the tracing probe, a gage-block must be used which has its sides ground and "perfectly" parallel to the axes of the machine.

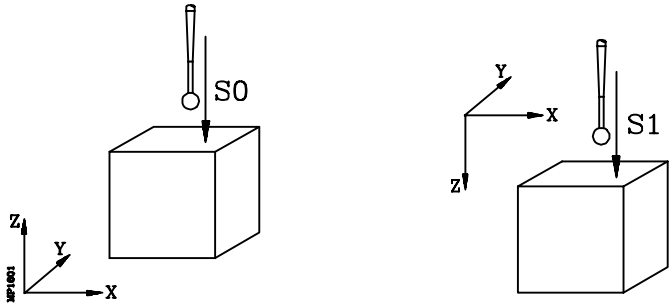
The CNC will treat the tracing probe as any other tool. Therefore, it must have its associated tool offset properly defined (probe length and ball radius).

Once the offset of the tracing probe has been selected, which must be installed on the longitudinal (perpendicular) axis, must be positioned over the center of the gage-block.

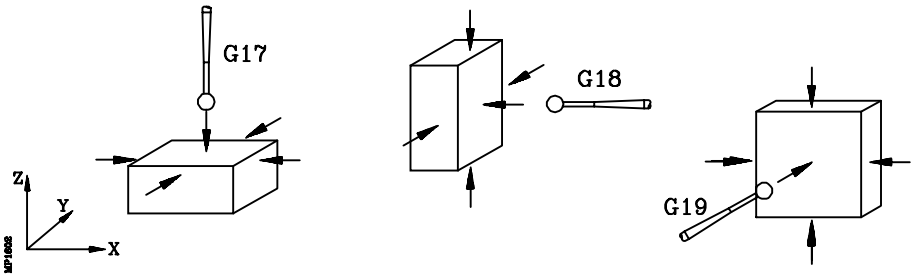
The programming format for this function is: **G26 S**

The **S** parameter indicates the direction of the part search along the perpendicular axis (carrying the probe). The possible values for this parameter are:

0 = Negative direction
1 = Positive direction



Once the probe makes contact with the surface of the gage-block, the CNC moves the probe on the surface measuring the rest of the sides as shown below:



Warning:

The feedrate for these movements must be selected before executing function G26.

The deviations of the probe along each axis X, Y, Z are stored internally to be used later as correction factors when executing a tracing operation (G23) or one of the tracing cycles (TRACE).

Whenever the display option for "Following Error" is selected in the JOG mode, the right-hand side of the CRT (in the window for probe values) will show the correction factor applied onto each axis, the deflections of each axis and the total deflection.

Function G26 is not modal. Therefore, it must be programmed every time the tracing probe is to be calibrated.

Nothing else may be programmed in the block defining function G26.

F A

16.3 G23. ACTIVATE TRACING

Once the tracing function is activated (G23), the CNC keeps the probe in contact with the surface of the model until this function is cancelled by G25.

When defining G23, it must be indicated the nominal deflection or pressure that the probe must keep while touching the surface of the model.

The types of tracing available with the G23 function are described next and they are:

* **Manual tracing.**

  The deflection of the probe depends on the pressure the operator exerts onto the probe.

* **One-dimensional tracing.**

  It is the most common type of tracing. The model sweeping axis must be defined. Once this type of tracing has been defined, the tracing path must be defined by means of the other two axes.

* **Two-dimensional tracing.**

  It contours the model. The two axes contouring the profile must be defined. Once this type of tracing has been defined, only the movements of the other axis can be programmed.

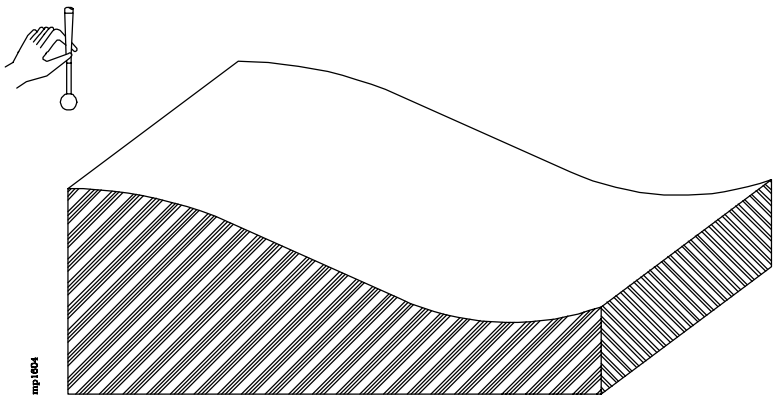
* **Three-dimensional tracing.**

  It contours the model. This profile contouring is carried out by the three axes. Therefore, all three of them must be defined. Once this type of tracing has been defined, it is not be possible to program the movements of the X, Y, and Z axes.

16.3.1 G23. ACTIVATE MANUAL TRACING

With this type of tracing, the operator may move the probe by hand on and along the surface of the model to be traced.

During this type of tracing, the deflection of the probe depends on the pressure that the operator exerts on to the probe. Therefore, it is advised to use this type of tracing for roughing operations or to use the digitizing function G24 so the CNC generates a program which compensates for the deflection of the probe.



Manual tracing must be selected in the MDI option of the JOG mode and the programming format is the following:

G23 [X] [Y] [Z]

X, Y, Z Define the axis or axes that will sweep the model.

It is possible to define one, two or three axes. When more than one axis is defined, they must be programmed in this order: X, Y, Z.

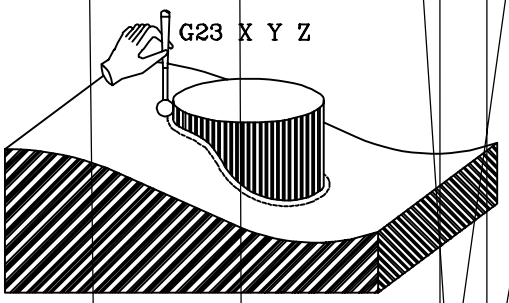
If no axis is defined, the CNC assumes the longitudinal (perpendicular) axis as the probing axis.

The probe will only be moved manually along the defined axes. The rest of the axes must be moved by means of the JOG keys, by using an electronic handwheel or by executing blocks in MDI mode.

For example. If the tracing function is activated as G23 Y Z, the probe may be moved by hand along the Y and Z axes. To move it along the X axis, either the JOG keys or an electronic handwheel must be used or execute blocks in MDI mode.

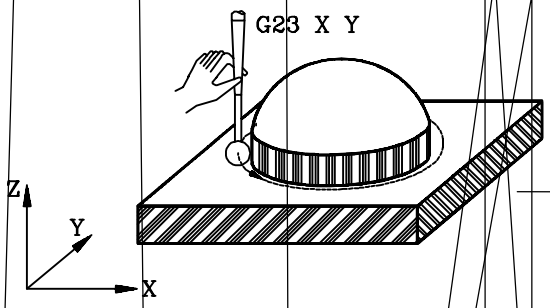
When trying to jog, or move with an electronic handwheel, one of the axes set as sweeping axes, the CNC will issue the corresponding error message.

Examples:



G23 X Y Z

* This option is very interesting to perform roughing operations or 3-D contouring.
* The operator may move the probe by hand in all directions.
* It is not possible to jog the X, Y, Z axes or move them with an electronic handwheel.



G23 X Y, G23 X Z , G23 YZ

* With this option it is possible to perform two-dimensional contouring or parallel tracing passes.
* The operator may move the probe by hand along the selected axes (Y and Z in the example of parallel tracing passes).
* It is only possible to move, by using the JOG keys or an electronic handwheel, the axis not selected (X in the example of parallel tracing passes).
* To make parallel tracing passes, the other axis must be moved by using the JOG keys or an electronic handwheel.

G23 X , G23 Y , G23 Z

* With this option it is possible to take (capture) data on specific points of the model.
* The operator may move the probe by hand only along the selected axis.
* The other two axes must be moved using the JOG keys or an electronic handwheel.ACTIVATE

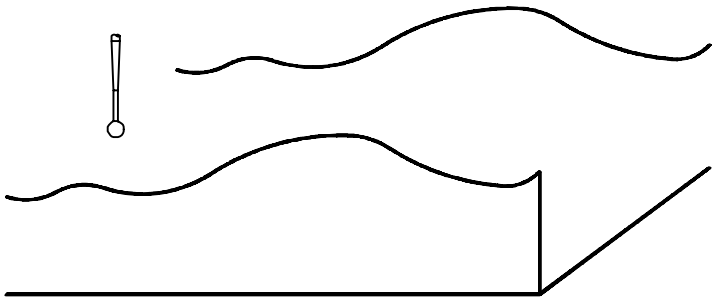
16.3.2 G23. ACTIVATE ONE-DIMENSIONAL TRACING

This type of tracing may be selected by part-program or in the MDI option the JOG and AUTOMATIC modes.

Once activated, the CNC will approach the probe to the model until it touches it and it maintains the probe in contact with the surface of the model at all times following the selected path.

The tracing path may be obtained either by programming it in ISO code or by moving the axes with the JOG keys or with an electronic handwheel.

It must be borne in mind that once this type of tracing has been activated, the sweeping axis may not be programmed or moved. If attempted to do so, the CNC will issue the corresponding error message.



The programming format is as follows:

G23 [axis] I±5.5 N5.5

[axis] Defines the axis sweeping the model.

It may be the X, Y or Z axis. If no axis is defined, the CNC assumes the longitudinal (perpendicular) axis as the sweeping axis.

The undefined axes must be used to define the tracing path either by programming it in ISO code or by moving them using the JOG keys or an electronic handwheel.

I±5.5 Defines the maximum tracing depth of the sweeping axis and it is referred to the position of the probe at the time it is being defined.

If part of the workpiece is out of this area (zone), the tracing function will assign to the sweeping axis the coordinate value of this parameter.

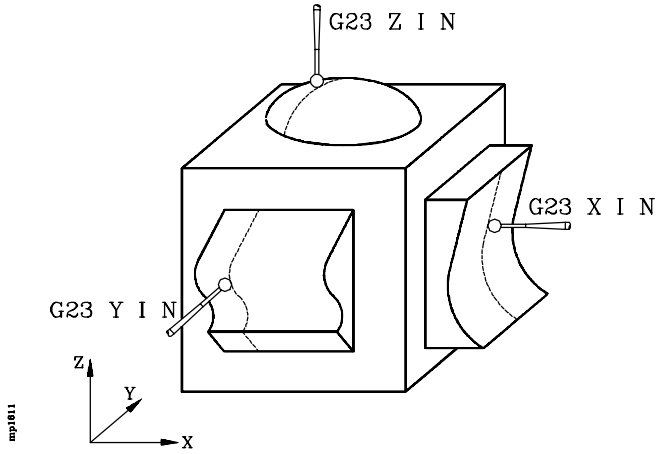
N 5.5 Nominal Deflection. Indicates the pressure kept by the probe while sweeping the surface of the model.

The deflection is given in the selected work units (mm or inches) and its value is usually comprised between 0.3mm and 1.5mm.

The tracing quality depends upon the amount of deflection being used, the tracing feedrate and the geometry of the model.

In order to prevent the probe from separating from the model, it is advised to use a profile tracing feedrate of about 1000 times the deflection value per minute. For example: for a deflection value of 1mm, the tracing feedrate would be 1m/min.

Application example on the X, Y and Z axes:



Programming example:

The tracing area is delimited between (X100 Y0) and (X150 Y50), the probe being on the Z axis.

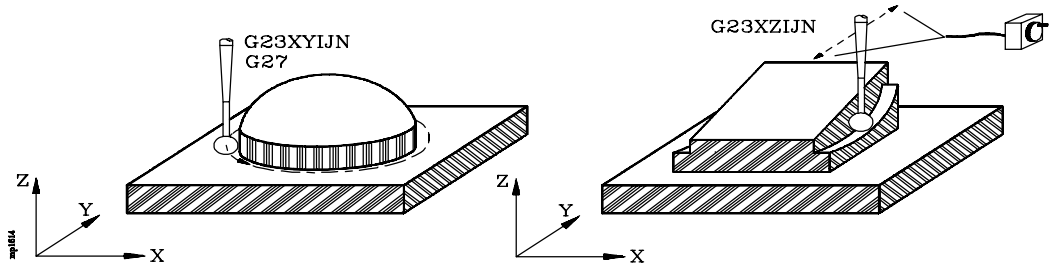
```
G90 G01 X100 Y0 Z80 F1000
G23  Z I-10 N1.2     Tracing ON
N10 G91 X50          Defines the sweep
Y5                        "
X-50                      "
N20 Y5                    "
(RPT N10,N20) N4 "
X50                       "
G25                  Tracing OFF
M30
```

16.3.3 G23. ACTIVATE TWO-DIMENSIONAL TRACING

With this type of tracing it is possible to perform two-dimensional contouring.

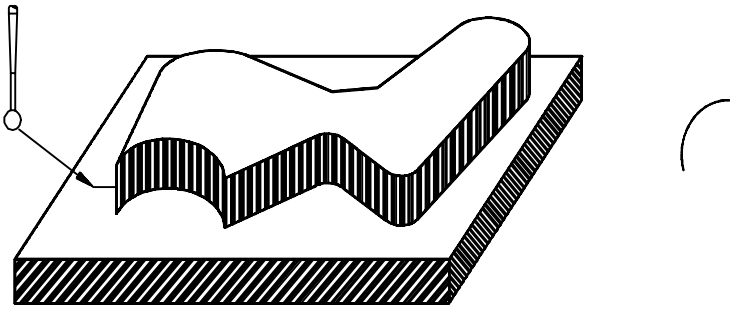
This type of tracing may be selected by part-program or in the MDI option the JOG and AUTOMATIC modes.

Once activated, the CNC will move the probe to the approach point (I,J) indicated when defining function G23. It then moves the probe until it touches the model and it maintains the probe in contact with the surface of the model at all times following the selected path.



It must be borne in mind that once this type of tracing has been activated, the sweeping axes may not be programmed or moved. If attempted to do so, the CNC will issue the corresponding error message.

The contouring path must be defined by means of function G27 (tracing contour definition) as described in this chapter or by moving the other axis (the one not following the profile) with the JOG keys or with an electronic handwheel.



The programming format is as follows:

G23 [axis1] [axis2] I±5.5 J±5.5 N5.5

axis1 axis2 Define the axes sweeping the model.

Two of the X, Y, and Z axes must be defined and in the indicated order.

I±5.5 Defines the approach coordinate for "axis1" and it is referred to part zero.

J±5.5 Defines the approach coordinate for "axis2" and it is referred to part zero.

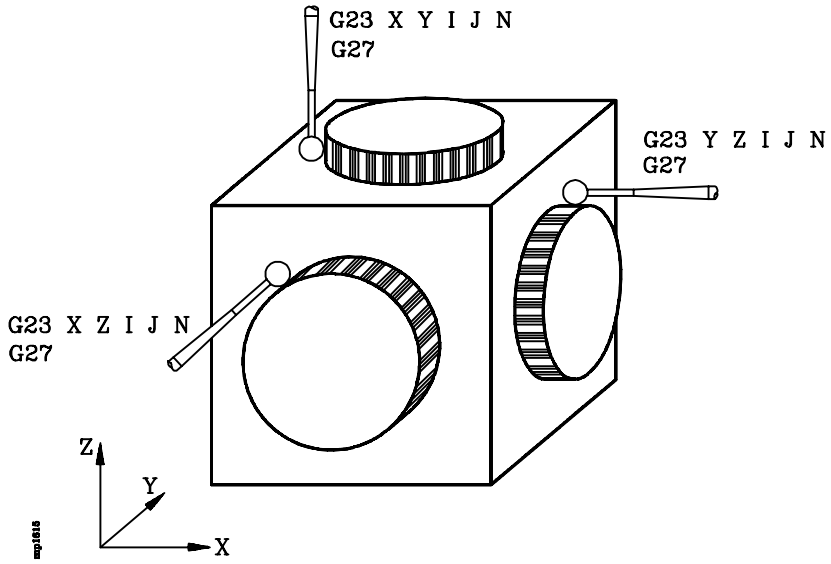
N 5.5 Nominal Deflection. Indicates the pressure kept by the probe while sweeping the surface of the model.

The deflection is given in the selected work units (mm or inches) and its value is usually comprised between 0.3mm and 1.5mm.

The tracing quality depends upon the amount of deflection being used, the tracing feedrate and the geometry of the model.

In order to prevent the probe from separating from the model, it is advised to use a profile tracing feedrate of about 1000 times the deflection value per minute. For example: for a deflection value of 1mm, the tracing feedrate would be 1m/min.

Tracing examples for various contours:



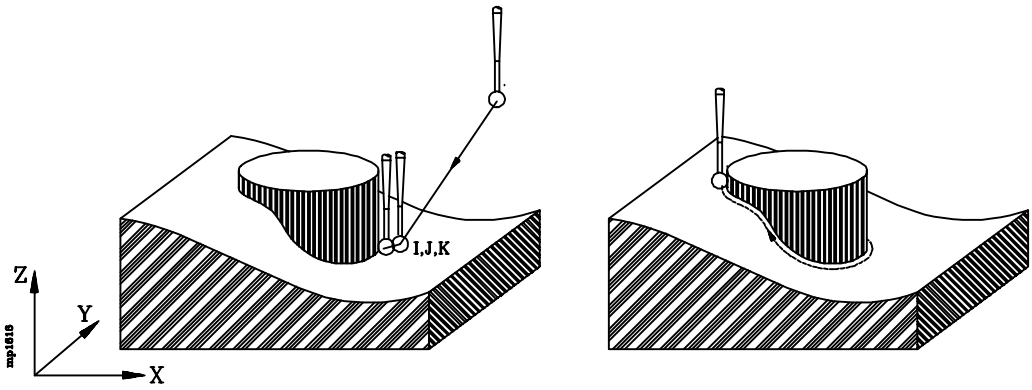
16.3.4 G23. ACTIVATE THREE-DIMENSIONAL TRACING

With this type of tracing it is possible to perform three-dimensional contouring.

There must always be a surface for the probe to touch. The maximum slope of this sweeping surface depends on the sweeping feedrate and the nominal deflections. The greater the sweeping feedrate the flatter the surface must be.

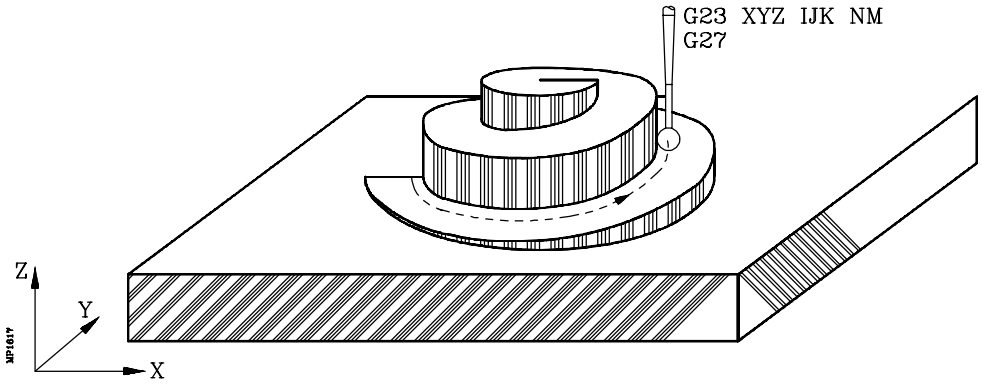
This type of tracing may be selected by part-program or in the MDI option the JOG and AUTOMATIC modes.

Once activated, the CNC will move the probe to the approach point (I,J,K) indicated when defining function G23. It then moves the probe until it touches the model and it maintains the probe in contact with the surface of the model at all times following the selected path.



It must be borne in mind that once this type of tracing has been activated, the sweeping (X,Y, Z) axes may not be programmed or moved. If attempted to do so, the CNC will issue the corresponding error message.

The contouring path must be defined by means of function G27 (tracing contour definition) as described in this chapter.



The programming format is as follows:

G23 X Y Z I±5.5 J±5.5 K ±5.5 N5.5 M5.5

X, Y, Z Define the axes sweeping the model.

All three axes (X, Y, and Z) must be defined and in this order.

I±5.5 Defines the approach coordinate for X and it is referred to part zero.

J±5.5 Defines the approach coordinate for Y and it is referred to part zero.

K±5.5 Defines the approach coordinate for Z and it is referred to part zero.

N5.5 Nominal deflection for the axes forming the plane.

M 5.5 Nominal Deflection for the longitudinal (perpendicular) axis.

The N and M deflection values indicate the pressure kept by the probe while sweeping the surface of the model.

The deflection is given in the selected work units (mm or inches) and its value is usually comprised between 0.3mm and 1.5mm.

The tracing quality depends upon the amount of deflection being used, the tracing feedrate and the geometry of the model.

In order to prevent the probe from separating from the model, it is advised to use a profile tracing feedrate of about 1000 times the deflection value per minute. For example: for a deflection value of 1mm, the tracing feedrate would be 1m/min.

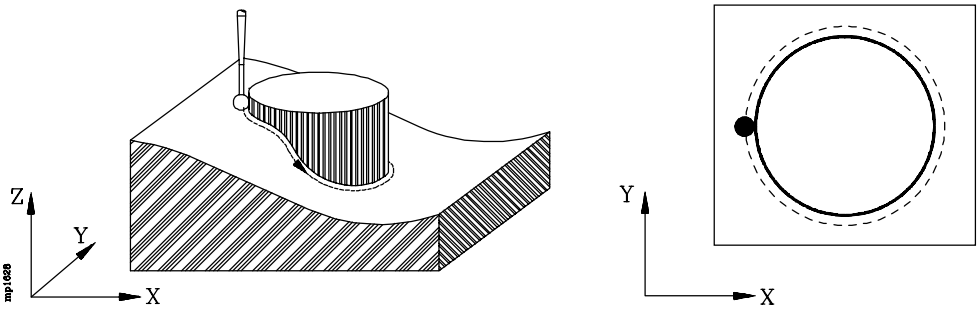
16.4 G27. TRACING CONTOUR DEFINITION

Whenever a two-dimensional or three-dimensional tracing function is activated, it is necessary to define the tracing contour by means of function G27.

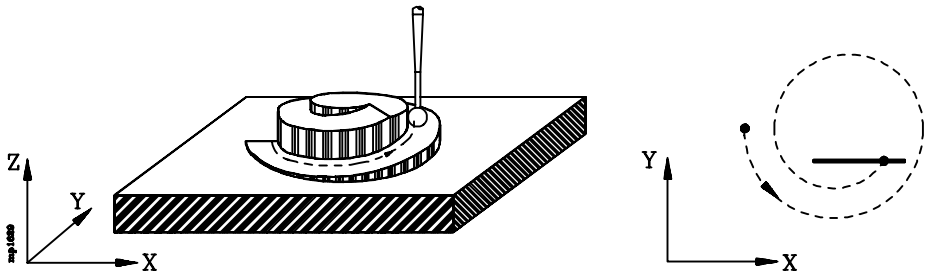
The tracing probe starts moving around the model keeping in constant contact with it in the indicated direction.

It is possible to define a closed contour (where the initial and final points are the same) or an open contour (where the initial and final points are not the same).

Example of a closed contour:



In the case of an open contour, it is necessary to define the end of the contour by means of a segment parallel to the axes. The tracing pass will end when the probe crosses this segment.

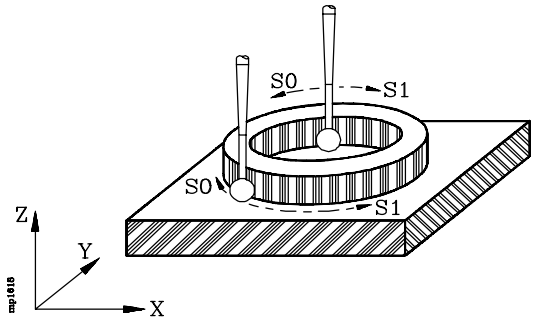


The programming format is the following:

G27 S Q±5.5 R±5.5 J5.5 K

S Indicates the direction of the sweep.

0 = The probe moves leaving the model to its right.
1 = The probe moves leaving the model to its left.



If not programmed, the CNC assumes a value of S0.

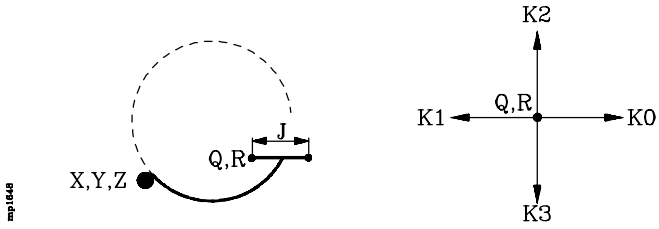
Q, R±5.5 These parameters must be set when defining an open contour (where the initial and final points are not the same).

They define the initial point of the segment that indicates the end of the contour. They must be referred to part zero. The Q coordinate corresponds to the abscissa axis and the R coordinate to the ordinate axis.

When defining a closed contour (where the initial and final points are the same), just program **G27 S**.

J 5.5 This parameter must be set when defining an open contour; that is, when Q and R have been defined.

It sets the length of the segment indicating the end of the contour.



K This parameter must be set when defining an open contour; that is, when Q and R have been defined.

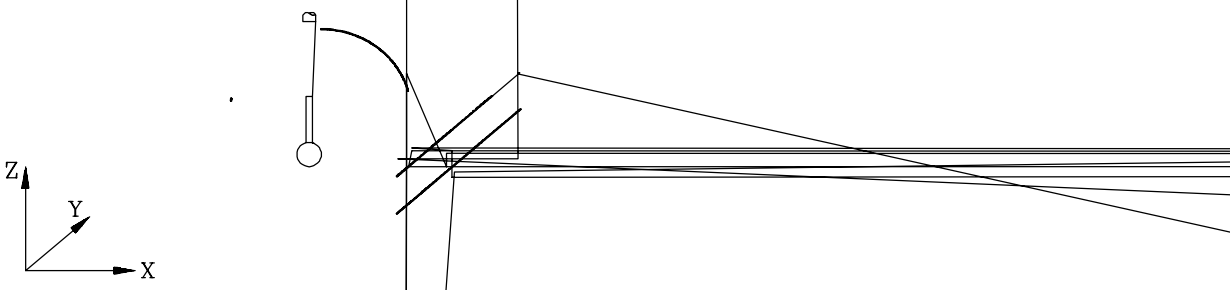
It sets the direction of the segment defining the end of the contour.

0 = Towards positive coordinate values of the abscissa axis.
1 = Towards negative coordinate values of the abscissa axis.
2 = Towards positive coordinate values of the ordinate axis.
3 = Towards negative coordinate values of the ordinate axis.

If not programmed, the CNC assumes a value of K0.

Two-dimensional programming examples:

Closed two-dimensional contour:




```
G23 XY I50 J8 N0.8      ;Two-dimensional tracing definition
G24 L8 E5 K1            ;Digitizing definition
G27 S0                  ;Closed contour definition
G25                     ;Deactivate tracing and digitizing
```

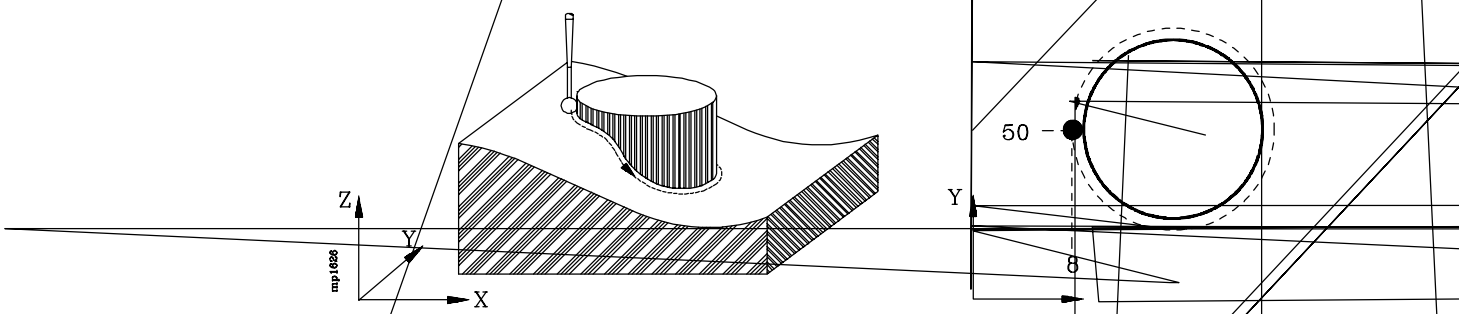
Open two-dimensional contour:

```
G23 XY I60 J20 N0.8     ;Two-dimensional tracing definition
G24 L8 E5 K1            ;Digitizing definition
G27 S0 Q10 R25 J15 K0   ;Open contour definition
G25                     ;Deactivate tracing and digitizing
```

TRACING CONTOURDEFIN

Three-dimensional programming examples:

Closed three-dimensional contour:




```
G23 XYZ I8 J50 K75 N0.8      ;Three-dimensional tracing definition
G24 L8 E5 K1                 ;Digitizing definition
G27 S1                       ;Closed contour definition
G25                          ;Deactivate tracing and digitizing
```

Open three-dimensional contour:

```
G23 XYZ I20 J50 K45 N0.8 M0.5    ;Three-dimensional tracing definition
G24 L8 E5 K1                     ;Digitizing definition
G27 S1 Q80 R40 J25 K0            ;Open contour definition
G25                              ;Deactivate tracing and digitizing
```

16.5 G25. DEACTIVATE TRACING

The tracing function can be cancelled (deactivated):

- By means of function G25 which can be programmed in any block.
- By selecting a new work plane (G16, G17, G18, G19).
- When selecting a new longitudinal (perpendicular) axis (G15).
- After executing an end of program (M02, M30).
- After an EMERGENCY or RESET.

When cancelling the tracing function (G23), the digitizing function (G24) will also be cancelled if it was active.

16.6 G24. ACTIVATE DIGITIZING

Digitizing consists in taking (capturing) coordinates of the machine while tracing the model and sending them to a file previously opened by the "OPEN P" statement.

Regardless of the type of the tracing being used (manual, one-dimensional, two-dimensional or three-dimensional) the digitized points show the coordinates along the X, Y and Z axes.

There are two types of digitizing: continuous and point-by-point.

Continuous Digitizing.

It may be used with any type of tracing. Its programming format is: **G24 L E K**

The CNC captures points of the model depending on the value assigned to parameters "L" and "E". If "L" is not programmed, the CNC will "understand" that a point-by-point digitizing is to be done.

Point-by-point Digitizing.

It may be used only when performing a manual tracing. That is, when the operator moves the probe by hand on and along the surface of the model.

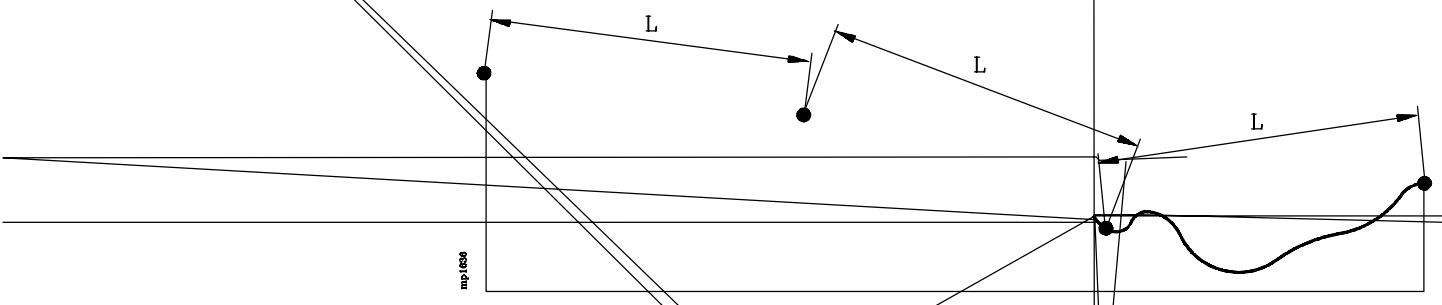
Its programming format is: **G24 K**

The CNC generates a new point whenever the operator presses the "READ POINT BY POINT" softkey or whenever the PLC provides an up-flank (leading edge) at the general logic input "POINT" of the CNC (external push-button).

The general programming format to activate the digitizing function is as follows:

G24 L5.5 E5.5 K

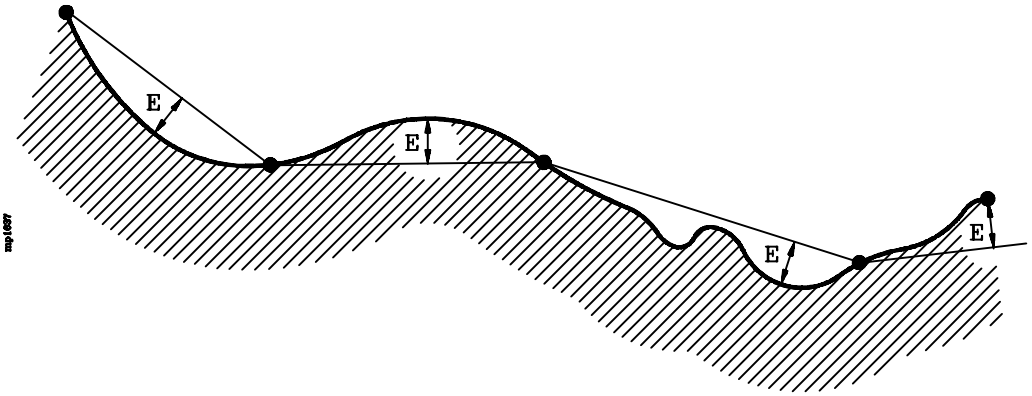
L 5.5 Indicates the sweeping step or distance between two consecutive digitized points.



The CNC provides the coordinates of a new point after moving, in space and along the programmed path, the distance indicated by parameter "L".

If not programmed, the CNC will "understand" that a point-by-point digitizing is to be done.

E 5.5 Indicates the chordal error or maximum difference allowed between the surface of the model and the segment joining two consecutive digitized points. It is given in the selected work units (millimeters or inches).



If not programmed or programmed with a value of "0", the chordal error will be ignored providing a new point after moving, in space and along the programmed path, the distance indicated by parameter "L".

K Indicates the storing format for the digitized points in the program selected by means of the "OPEN P" statement.

K=0 Absolute format.

All points will be programmed in absolute coordinates (G90) and defined by the X, Y and Z axes.

K=1 Absolute filtered format.

All points will be programmed in absolute coordinates (G90); but only those axes whose positions have changed with respect to the previous digitized point will be defined.

K=2 Incremental filtered format.

All points will be programmed in incremental coordinates (G91) and referred to the previous digitized point. Only those axes whose positions have changed with respect to the previous digitized point will be defined.

If not programmed, the canned cycle will assume a value of K0.

Basic concepts:

* Function G24 must be defined just before the block where the digitizing begins.

* Before activating the digitizing function (G24), it is necessary to open, by means of the "OPEN P" statement, the program which will store the digitized points.

If instead of storing the digitized points in a CNC program, it is desired to store them in a peripheral device or PC via DNC, it must be so indicated when defining the "OPEN P" statement.

When communicating via DNC, if the data transmission rate is lower than the data acquisition (capture) speed, the resulting tracing operation will be slower.

* During the digitizing of the model, the CNC only controls the movements of the X, Y, Z axes. Therefore, the generated program blocks will only contain some or all of these axes.

* No points will be generated while the probe is seeking the model or when it is off the surface of the model.

* The CNC takes into account the deflections of the probe when calculating the coordinates of the new digitized point.

* To deactivate the digitizing function, program G25.

  The digitizing function is also cancelled (deactivated) when deactivating the tracing function (G23) and, consequently, in the following instances:

  - When selecting a new work plane (G16, G17, G18, G19).

  - When selecting a new longitudinal (perpendicular) axis (G15).

  - After executing an end of program (M02, M30).

  - After an EMERGENCY or RESET.

Programming example:

G17	Selects the Z axis as longitudinal (perpendicular)
G90 G01 X65 Y0 F1000	Positioning
(OPEN P12345)	Program receiving (storing) digitized data
(WRITE G01 G05 F1000)	
G23 Z I-10 N1	Tracing ON
G24 L8 E5 K1	Digitizing ON
G1 X100 Y35	Define tracing path
"	"
"	"
G25	Cancel tracing and digitizing
M30	

16.7 TRACING AND DIGITIZING CANNED CYCLES

The tracing/digitizing canned cycles offered by this CNC are based on the types of tracing described earlier and they are the following:

TRACE 1 Tracing / digitizing in a grid pattern
TRACE 2 Tracing / digitizing in an arc pattern.
TRACE 3 Profile tracing / digitizing in the plane
TRACE 4 3-D Profile tracing / digitizing (in space)
TRACE 5 Tracing / digitizing with polygonal sweep.

They are programmed by means of the high level instruction **TRACE.** The cycle number may be indicated either by a number (1, 2, 3, 4, 5) or by an expression whose result is one of these numbers.

They all have a series of parameters defining the tracing path and the digitizing conditions.

To just trace the part without digitizing it, the digitizing parameters must be set to "0".

To digitize the model, besides setting the digitizing parameters, the following points must be considered:

* Before calling the canned cycle, it is required to open the program which will store the digitized data by means of the "**OPEN P**" statement.

* If the captured data is supposed to be stored at a peripheral device or computer via DNC instead of doing it at the CNC part-memory, it must be so indicated when defining the "**OPEN P**" statement.

* It must be borne in mind that the generated program blocks are positioning only (G01 X Y Z). Therefore, it is convenient to also include in such program the machining conditions by using the "**WRITE**" statement.

* Once the digitizing process is over, an end-of-program (M02 or M30) must also be written by means of the "**WRITE**" statement.

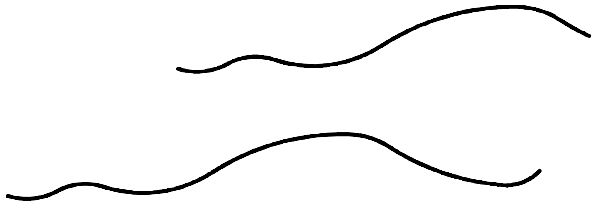
Once the tracing cycle is over, the probe will be positioned where it was before executing the cycle.

The execution of a tracing canned cycle does not change the history of the previous "G" functions.

16.7.1 GRID -PATTERN TRACING CANNED CYCLE

The programming format for this cycle is as follows:

(TRACE 1, X, Y, Z, I, J, K, A, C, Q, D, N, L, E, G, H, F)



X±5.5 Theoretical absolute coordinate value along the abscissa axis of the first probing point. It must coincide with one of the corners of the grid.

Y±5.5 Theoretical absolute coordinate value along the ordinate axis of the first probing point. It must coincide with one of the corners of the grid.

Z±5.5 Theoretical coordinate value along the probing axis (longitudinal/perpendicular) where the probe is to be positioned before starting the tracing operation.

It is given in absolute values and it must be off the model maintaining a safety distance from its outermost surface.

I±5.5 Defines the maximum tracing depth and it is referred to the coordinate value given to parameter Z.

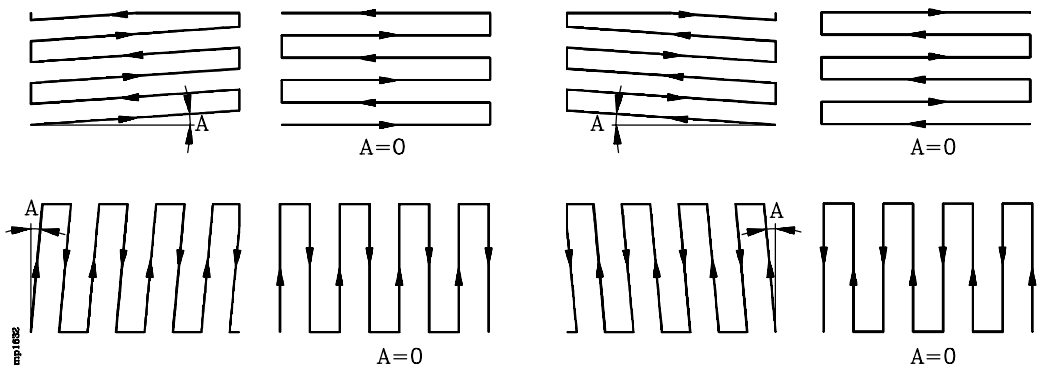
If part of the model is out of this area, the tracing will assign this maximum depth to the probing axis and will continue executing the tracing cycle without issuing an error.

If programmed with a value of "0", the CNC will issue the corresponding error.

J±5.5 Defines the length of the grid along the abscissa axis. The positive sign indicates that the grid is located to the right of the point (X, Y) and the negative sign that it is to the left of that point.

K±5.5 Defines the length of the grid along the ordinate axis. The positive sign indicates that the grid is located above the point (X, Y) and the negative sign that it is below that point.

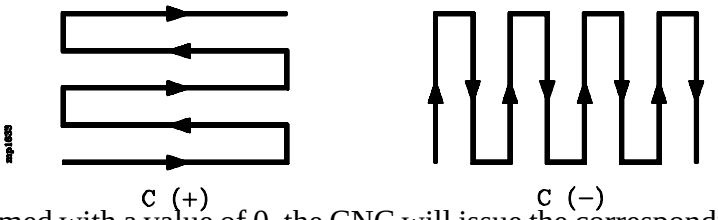
A 5.5 Defines the angle of the sweeping path.



It must be comprised between 0º (included) and 90º (not included). If not programmed, the canned cycle will assume a value of "A0".

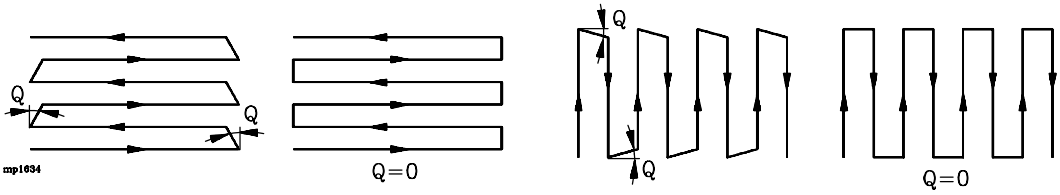
C±5.5 Defines the distance which will be maintained between two tracing passes.

If programmed with a positive value, the tracing operation will be carried out along the abscissa axis and the distance will be taken along the ordinate axis. On the other hand, if programmed with a negative value, the tracing operation will be carried out along the ordinate axis and the distance will be taken along the abscissa axis.



If programmed with a value of 0, the CNC will issue the corresponding error.

Q 5.5 Defines the angle of the incremental path.



It must be comprised between 0º and 45º (both included). If not programmed or if an one-directional tracing is programmed (D=1), the canned cycle will assume a value of "Q0".

D Indicates how the grid is followed according to this code:

0 = The tracing is carried out in both directions (zig-zag).
1 = The tracing is carried out following the grid in one direction.

If not programmed, the canned cycle assumes a value of "D0".

N 5.5 Nominal Deflection. Indicates the pressure kept by the probe while sweeping the surface of the model.

The deflection is given in the selected work units (mm or inches) and its value is usually comprised between 0.3mm and 1.5mm.

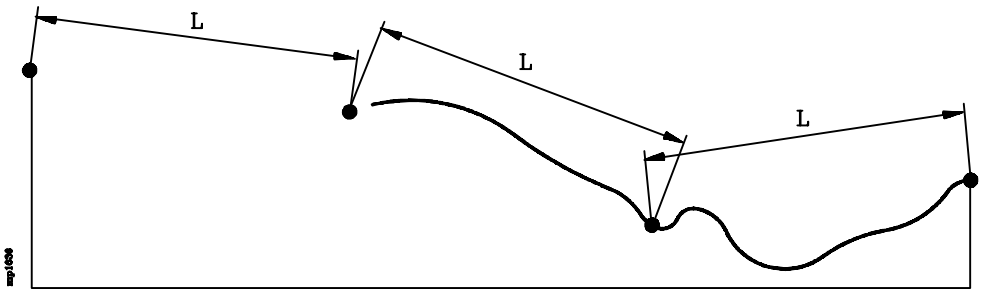
The tracing quality depends upon the amount of deflection being used, the tracing feedrate and the geometry of the model.

In order to prevent the probe from separating from the model, it is advised to use a profile tracing feedrate of about 1000 times the deflection value per minute. For example: for a deflection value of 1mm, the tracing feedrate would be 1m/min.

If not programmed, the canned cycle will assume a value of 1mm (0.03937").

L 5.5 This parameter must be defined when digitizing a part besides tracing it.

It indicates the sweeping step or distance between two consecutive digitized points.



The CNC keeps the probe in constant contact with the surface of the model and it provides the coordinates of a new point after moving, in space and along the programmed path, the distance indicated by parameter "L".

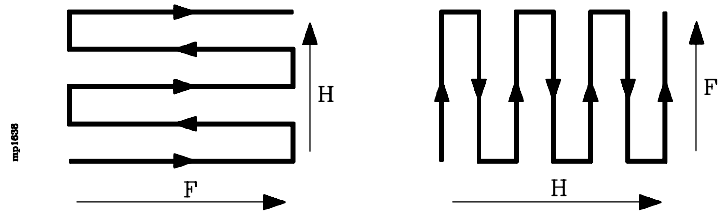
If not programmed or programmed with a value of "0", the canned cycle will assume that the model is **not** to bogragitized.

G=2 Incremental filtered format.

All points will be programmed in incremental coordinates (G91) and referred to the previous digitized point. Only those axes whose positions have changed with respect to the previous digitized point will be defined.

If not programmed, the canned cycle will assume a value of G0.

H5.5 Defines the feedrate for the incremental paths. It is programmed in mm/min. or inches/min.



If not programmed, the canned cycle will assume the "F" value (feedrate for the sweeping paths).

F5.5 Defines the sweeping feedrate. It is programmed in mm/min. or inches/min.

BASIC OPERATION:

1.- The probe positions at the point set by parameters X, Y and Z.

2.- The CNC approaches the probe to the model until it touches it.

3.- The probe keeps in constant contact with the surface of the model following it along the programmed path.

If it is to be digitized, (parameters "L" and "E") it will generate a new block per every digitized point in the program previously opened by means of the "**OPEN P**" statement.

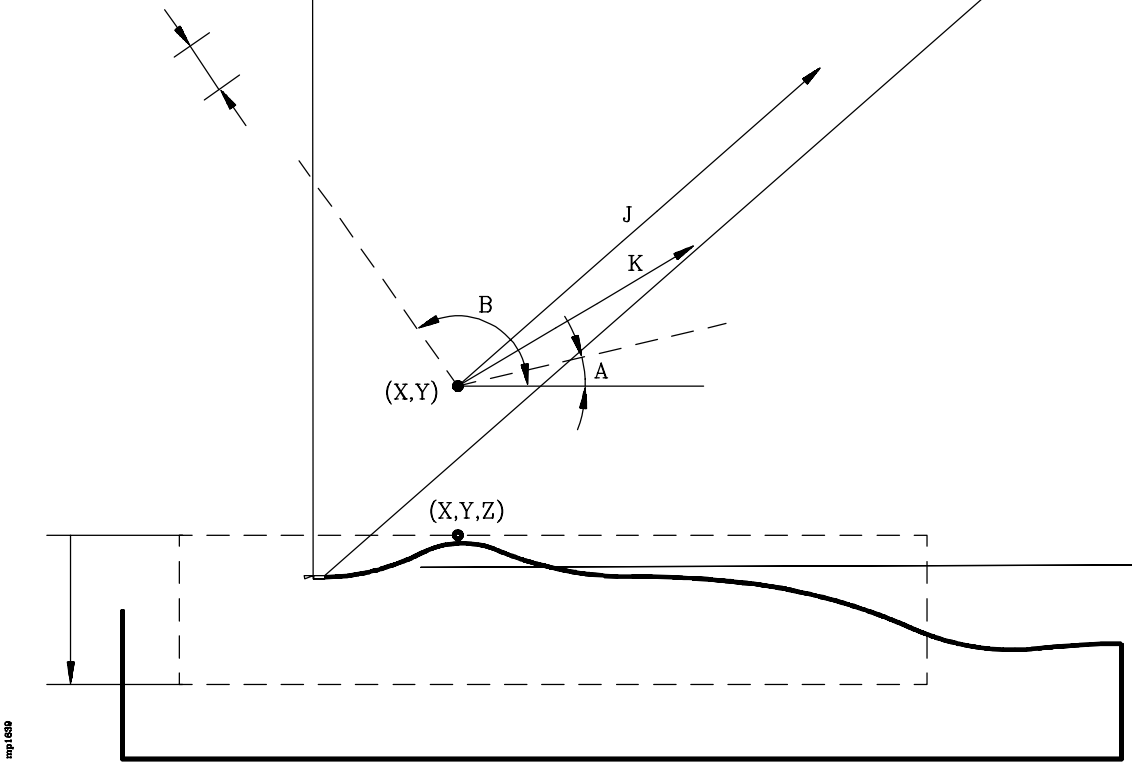
4.- Once the canned cycle is finished, the probe will return to the cycle calling point. This move consists of:

* Movement of the probe along the probing axis.
* Movement of the probe in the main work plane.

16.7.2 ARC PATTERN TRACING CANNED CYCLE

The programming format for this cycle is as follows:

(TRACE 2, X, Y, Z, I, J, K, A, B, C, D, R, N, L, E, G, H, F)



X±5.5 Theoretical absolute coordinate of the arc center along the abscissa axis.

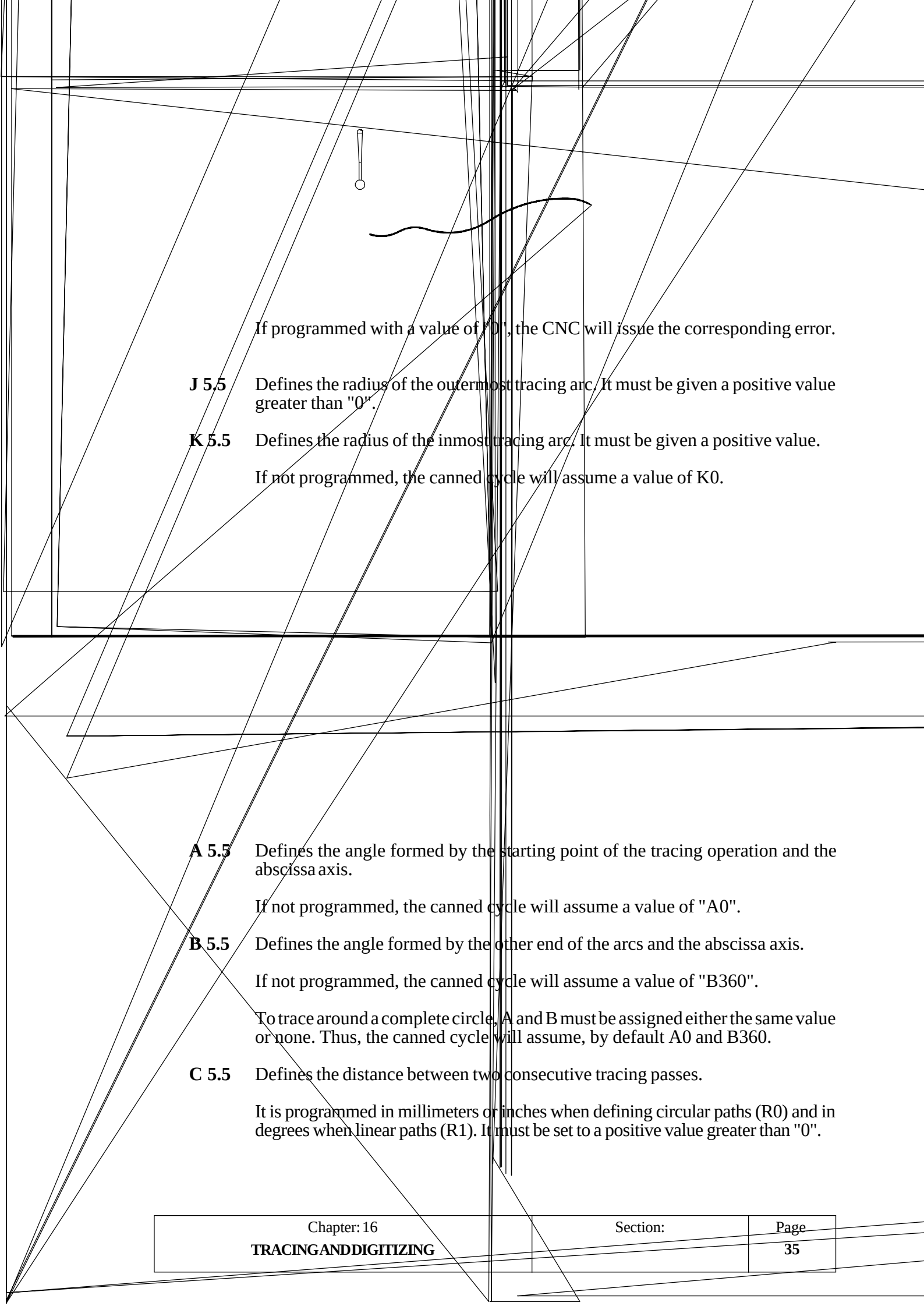
Y±5.5 Theoretical absolute coordinate of the arc center along the ordinate axis.

Z±5.5 Theoretical coordinate along the probing axis (longitudinal / perpendicular) where the probe is to be positioned before starting the tracing operation.

It is given in absolute values and it must be off the model at a safety distance from its outermost surface.

I±5.5 Defines the maximum tracing depth and it is referred to the coordinate value given to parameter Z.

If part of the model is out of this area, the tracing will assign this maximum depth to the probing axis and will continue executing the tracing cycle without issuing an error.



If programmed with a value of "0", the CNC will issue the corresponding error.

J 5.5 Defines the radius of the outermost tracing arc. It must be given a positive value greater than "0".

K 5.5 Defines the radius of the inmost tracing arc. It must be given a positive value.

If not programmed, the canned cycle will assume a value of K0.

A 5.5 Defines the angle formed by the starting point of the tracing operation and the abscissa axis.

If not programmed, the canned cycle will assume a value of "A0".

B 5.5 Defines the angle formed by the other end of the arcs and the abscissa axis.

If not programmed, the canned cycle will assume a value of "B360".

To trace around a complete circle, A and B must be assigned either the same value or none. Thus, the canned cycle will assume, by default A0 and B360.

C 5.5 Defines the distance between two consecutive tracing passes.

It is programmed in millimeters or inches when defining circular paths (R0) and in degrees when linear paths (R1). It must be set to a positive value greater than "0".

D Indicates how the sweep is performed according to the following code:

0 = The sweep is carried out in both directions (zig-zag).
1 = The sweep is always carried out in one direction).

If not programmed, the canned cycle assumes a value of "0".

R Indicates the type of sweeping path to be used according to the following code:

0 = Circular path, along the arc.
1 = Linear path, along the radius.

If not programmed, the canned cycle assumes a value of "0".

When selecting R0 (circular path):

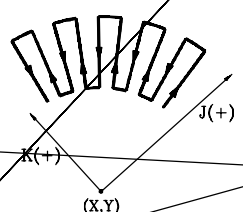
* When defining parameters A and B, it must be borne in mind that the first sweep is always done counter-clockwise.

* The step C indicates the linear distance between every two consecutive passes. It must be programmed in millimeters or inches.

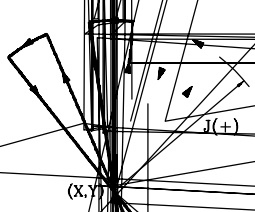
When selecting R1 (linear path):

* The step C indicates the angular distance between two consecutive passes. It must be programmed in degrees.

* Parameter K, inmost arc radius, may be programmed with either positive or negative values.



R1 D0



R1 D0

* If R1 D1 is selected (unidirectional linear path) the sweep will always be carried out from the inmost radius (K) to the outermost one (J).

N 5.5 Nominal Deflection. Indicates the pressure kept by the probe while sweeping the surface of the model.

The deflection is given in the selected work units (mm or inches) and its value is usually comprised between 0.3mm and 1.5mm.

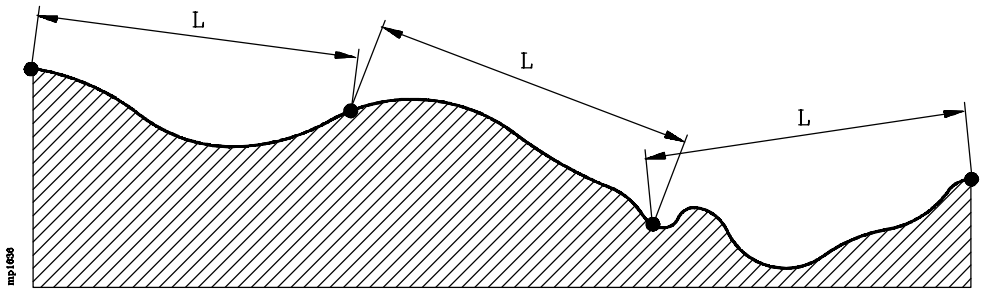
The tracing quality depends upon the amount of deflection being used, the tracing feedrate and the geometry of the model.

In order to prevent the probe from separating from the model, it is advised to use a profile tracing feedrate of about 1000 times the deflection value per minute. For example: for a deflection value of 1mm, the tracing feedrate would be 1m/min.

If not programmed, the canned cycle will assume the value of 1mm (0.03937").

L 5.5 This parameter must be defined when digitizing a part besides tracing it.

It indicates the sweeping step of distance between two consecutive digitized points.



The CNC keeps the probe in constant contact with the surface of the model and it provides the coordinates of a new point after moving, in space and along the programmed path, the distance indicated by parameter "L".

If not programmed or programmed with a value of "0", the canned cycle will assume that the model is **not** to be digitized.

E 5.5 This parameter must be defined when digitizing a part besides tracing it.

It indicates the chordal error or maximum difference allowed between the surface of the model and the segment joining two consecutive digitized points. It is given in the selected work units (millimeters or inches).

Page	Chapter: 16	Section:
G=0 Absolute format G=1 Absolute filtered format G=2 Incremental filtered format..		

If not programmed, the canned cycle will assume the F value (feedrate for the sweeping paths).

F5.5 Defines the sweeping feedrate. It is given in mm/min or inches/min.

BASIC OPERATION:

1.- The probe positions at the point set by parameters X, Y and Z.

2.- The CNC approaches the probe to the model until it touches it.

3.- The probe keeps in constant contact with the surface of the model following it along the programmed path.

If it is to be digitized, (parameters "L" and "E") it will generate a new block per every digitized point in the program previously opened by means of the "**OPEN P**" statement.

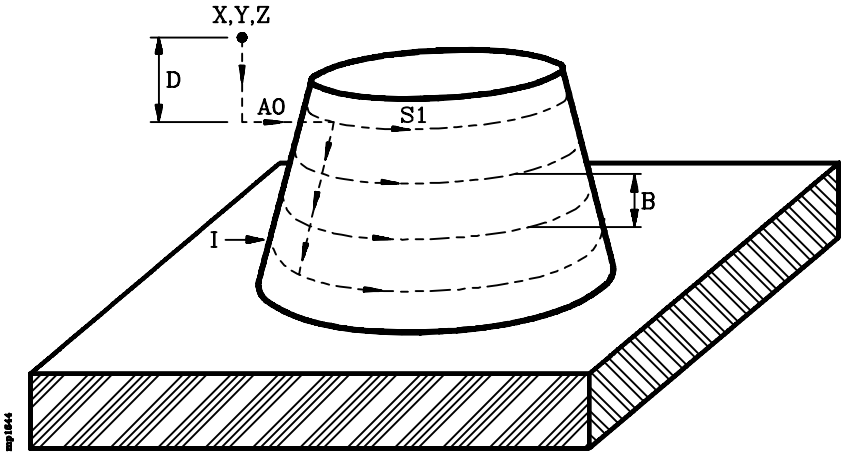
4.- Once the canned cycle is finished, the probe will return to the cycle calling point. This move consists of:

* Movement of the probe along the probing axis.
* Movement of the probe in the main work plane.

16.7.3 PROFILE TRACING CANNED CYCLE ALONG A PLANE

The programming format for this cycle is as follows:

(TRACE 3, X, Y, Z, I, D, B, A, C, S, Q, R, J, K, N, L, E, G, H, F)



X±5.5 Absolute theoretical coordinate value along the abscissa axis of the approach point. It must be off the model.

Y±5.5 Absolute theoretical coordinate value along the ordinate axis of the approach point. It must be off the model.

Z±5.5 Absolute theoretical coordinate value along the probing axis (longitudinal / perpendicular) where the probe is to be positioned before starting the tracing operation.

It must be off the model at a safety distance from its outermost surface.

I±5.5 Theoretical coordinate value along the probing axis (longitudinal / perpendicular) where the final tracing pass will be carried out.

D 5.5 Defines, along the probing axis, the distance between the "Z" position of the probe (described above) and the plane where the first tracing pass will be carried out.

If not programmed, the CNC will only make one tracing pass at the height indicated by parameter "I".

B 5.5 This parameters must be defined whenever parameter "D" is defined.

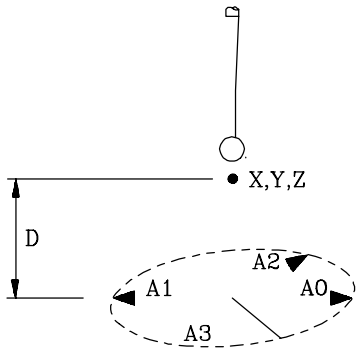
Defines, along the probing axis, the distance between two consecutive tracing passes.

If programmed with a value of "0", the CNC will issue the corresponding error.

A Indicates the tracing direction of the probe after positioning at X Y Z and having come down to the plane where the first tracing pass will be carried out seeking the model.

0 = Towards positive abscissa coordinates
1 = Towards negative abscissa coordinates
2 = Towards positive ordinate coordinates
3 = Towards negative ordinate coordinates

If not programmed, the CNC assumes A0



C This parameter is related to parameter A.

It indicates the maximum distance the probe may move to find the model.

S Indicates the direction used to trace the model.

0 = The probe moves leaving the model to its right.
1 = The probe moves leaving the model to its left.

If not programmed, the CNC assumes a value of "S0".

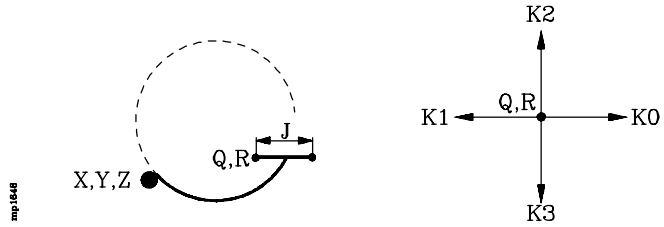
Q, R±5.5 These parameters must be defined when the contour is **not** closed.

Define the initial point of the segment which indicates the end of the contour. They are referred to part zero. The "Q" coordinate corresponds to the abscissa axis and the "R" to the ordinate axis.

If these parameters are not defined, the CNC performs the tracing of a closed contour. (Figure on the left).**PLANE PROFILE TRACINGCANNED CYCLE**

J 5.5 This parameter must be defined when the contour is **not** closed. In other words, when "Q" and "R" have been defined.

It defines the length of the segment which indicates the end of the contour.



If not programmed, the CNC assumes an infinite value

K This parameter must be defined when the contour is **not** closed. In other words, when "Q" and "R" have been defined.

It defines the direction of the segment which indicates the end of the contour.

0 = Towards positive abscissa coordinates
1 = Towards negative abscissa coordinates
2 = Towards positive ordinate coordinates
3 = Towards negative ordinate coordinates

If not programmed, the CNC assumes K0

N 5.5 Nominal Deflection. Indicates the pressure kept by the probe while sweeping the surface of the model.

The deflection is given in the selected work units (mm or inches) and its value is usually comprised between 0.3mm and 1.5mm.

The tracing quality depends upon the amount of deflection being used, the tracing feedrate and the geometry of the model.

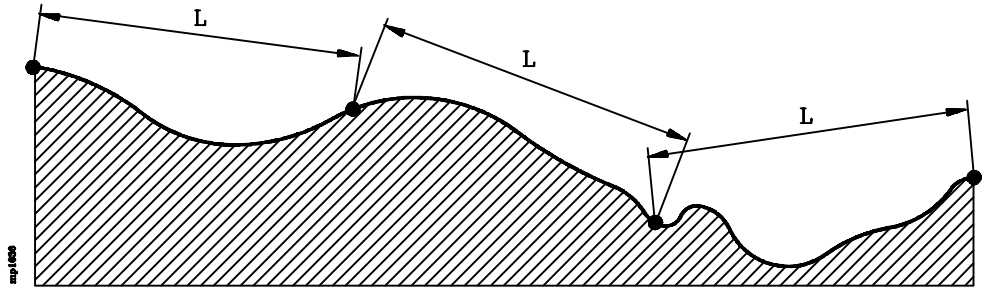
In order to prevent the probe from separating from the model, it is advised to use a profile tracing feedrate of about 1000 times the deflection value per minute. For example: for a deflection value of 1mm, the tracing feedrate would be 1m/min.

If not programmed, the canned cycle will assume the value of 1mm (0.03937").

L 5.5 This parameter must be defined when digitizing a part besides tracing it.

It indicates the sweeping step of distance between two consecutive digitized points.

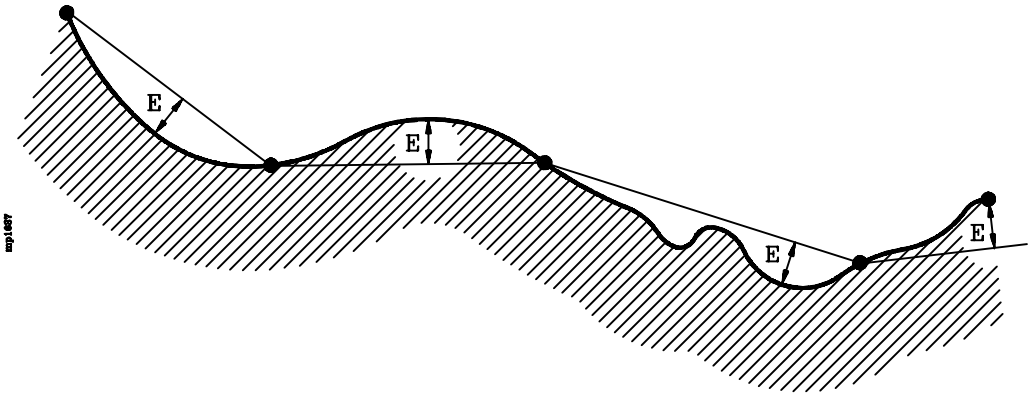
The CNC keeps the probe in constant contact with the surface of the model and it provides the coordinates of a new point after moving, in space and along the programmed path, the distance indicated by parameter "L".



If not programmed or programmed with a value of "0", the canned cycle will assume that the model is **not** to be digitized.

E 5.5 This parameter must be defined when digitizing a part besides tracing it.

It indicates the chordal error or maximum difference allowed between the surface of the model and the segment joining two consecutive digitized points. It is given in the selected work units (millimeters or inches).



If not programmed or programmed with a value of "0", the chordal error will be ignored and a new point will be provided after moving the "L" distance in space and along the programmed path.

G This parameter must be defined when digitizing the model besides tracing it.

Indicates the storing format for the digitized points in the program selected by means of the "**OPEN P**" statement.

G=0 Absolute format.

All points will be programmed in absolute coordinates (G90) and defined by the X, Y and Z axes.

G=1 Absolute filtered format..

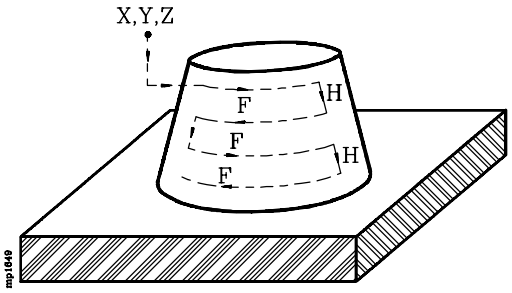
All points will be programmed in absolute coordinates (G90); but only those axes whose positions have changed with respect to the previous digitized point will be defined.

G=2 Incremental filtered format..

All points will be programmed in incremental coordinates (G91) and referred to the previous digitized point. Only those axes whose positions have changed with respect to the previous digitized point will be defined.

If not programmed, the canned cycle will assume a value of G0.

H5.5 Defines the feedrate for the incremental paths. It is programmed in mm/min. or inches/min.



If not programmed, the canned cycle will assume the F value (feedrate for the sweeping paths).

F5.5 Defines the sweeping feedrate. It is given in mm/min or inches/min.

BASIC OPERATION:

1.- The probe positions at the point set by parameters X, Y and Z.

2.- The CNC approaches the probe to the model until it touches it.

3.- The probe keeps in constant contact with the surface of the model following it along the programmed path.

If it is to be digitized, (parameters "L" and "E") it will generate a new block per every digitized point in the program previously opened by means of the "**OPEN P**" statement.

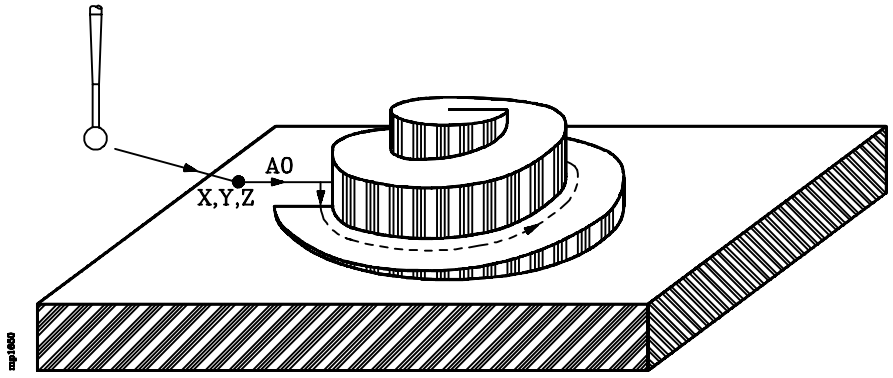
4.- Once the canned cycle is finished, the probe will return to the cycle calling point. This move consists of:

* Movement of the probe along the probing axis.
* Movement of the probe in the main work plane.

16.7.4 3-D PROFILE TRACING CANNED CYCLE

The programming format for this cycle is as follows:

(TRACE 4, X, Y, Z, I, A, C, S, Q, R, J, K, M, N, L, E, G, F)



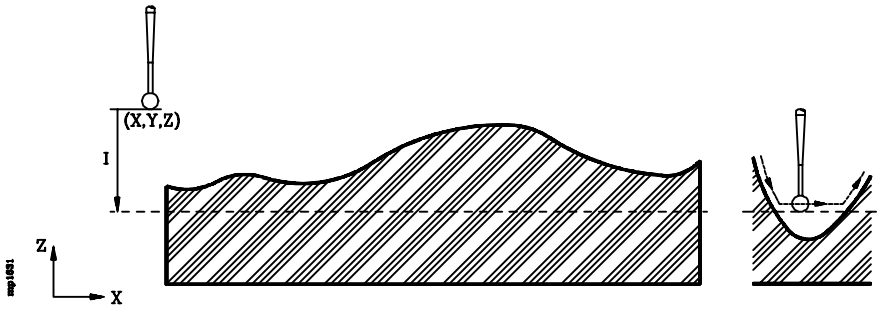
X±5.5 Absolute theoretical coordinate value along the abscissa axis of the approach point. It must be off the model.

Y±5.5 Absolute theoretical coordinate value along the ordinate axis of the approach point. It must be off the model.

Z±5.5 Absolute theoretical coordinate value along the probing axis (longitudinal / perpendicular) of the approach point.

It must be off the model and over it since the first movement, to seek the model, is carried out in the work plane.

I±5.5 Defines the maximum tracing depth and it is referred to the coordinate value given to parameter Z.

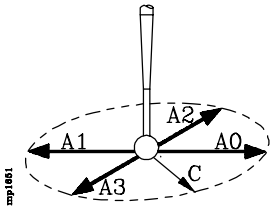


If part of the model is out of this area, the tracing will assign this maximum depth to the probing axis and will continue executing the tracing cycle without issuing an error.

If programmed with a value of "0", the CNC will issue the corresponding error.

A Indicates the tracing direction of the probe after positioning at X Y Z and having come down to the plane where the first tracing pass will be carried out seeking the model.

0 = Towards positive abscissa coordinates
1 = Towards negative abscissa coordinates
2 = Towards positive ordinate coordinates
3 = Towards negative ordinate coordinates



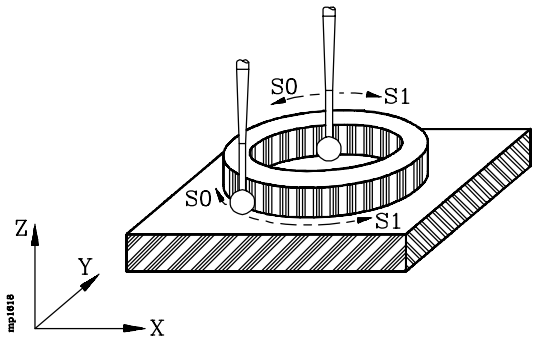
If not programmed, the CNC assumes A0.

C This parameter is related to parameter A.

It indicates the maximum distance the probe may move to find the model.

S Indicates the direction used to trace the model.

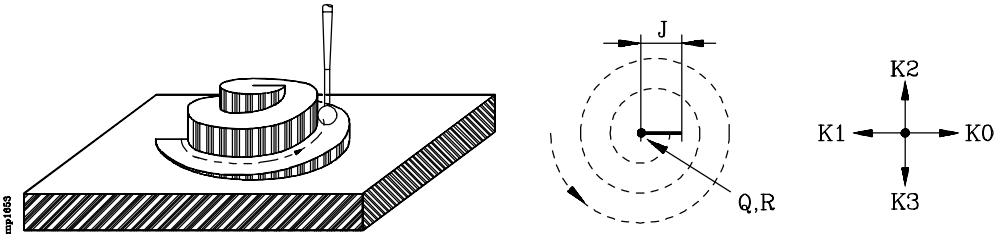
0 = The probe moves leaving the model to its right.
1 = The probe moves leaving the model to its left.



If not programmed, the CNC assumes a value of "S0".

Q, R±5.5 These parameters must be defined when the contour is **not** closed.

Define the initial point of the segment which indicates the end of the contour. They are referred to part zero. The "Q" coordinate corresponds to the abscissa axis and the "R" to the ordinate axis.



J 5.5 This parameter must be defined when the contour is **not** closed. In other words, when "Q" and "R" have been defined.

It defines the length of the segment which indicates the end of the contour.
If not programmed, the CNC assumes an infinite value.

K This parameter must be defined when the contour is **not** closed. In other words, when "Q" and "R" have been defined.

It defines the direction of the segment which indicates the end of the contour.

0 = Towards positive abscissa coordinates
1 = Towards negative abscissa coordinates
2 = Towards positive ordinate coordinates
3 = Towards negative ordinate coordinates

If not programmed, the CNC assumes K0.

M5.5 Nominal deflection of the probing axis (longitudinal / perpendicular)

If not programmed, the canned cycle will assume the value of 1mm (0.03937").

N 5.5 Nominal deflection of the axes forming the plane.

"**M**" and "**N**" indicate the pressure kept by the probe while sweeping the surface of the model.

The deflection is given in the selected work units (mm or inches) and its value is usually comprised between 0.3mm and 1.5mm.

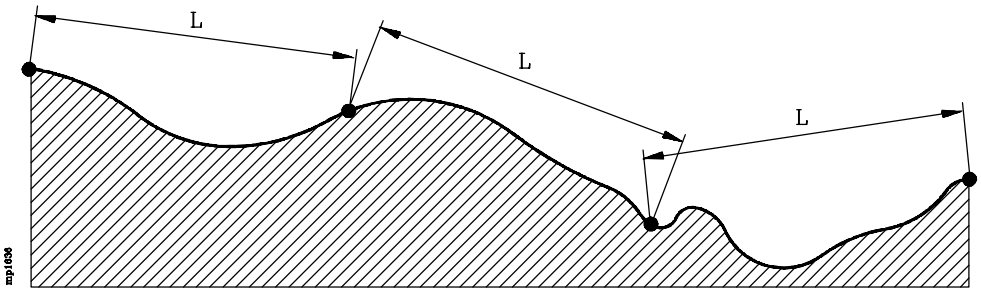
The tracing quality depends upon the amount of deflection being used, the tracing feedrate and the geometry of the model.

In order to prevent the probe from separating from the model, it is advised to use a profile tracing feedrate of about 1000 times the deflection value per minute. For example: for a deflection value of 1mm, the tracing feedrate would be 1m/min.

If not programmed, the canned cycle will assume the value of 1mm (0.03937").

L 5.5 This parameter must be defined when digitizing a part besides tracing it.

It indicates the sweeping step of distance between two consecutive digitized points.

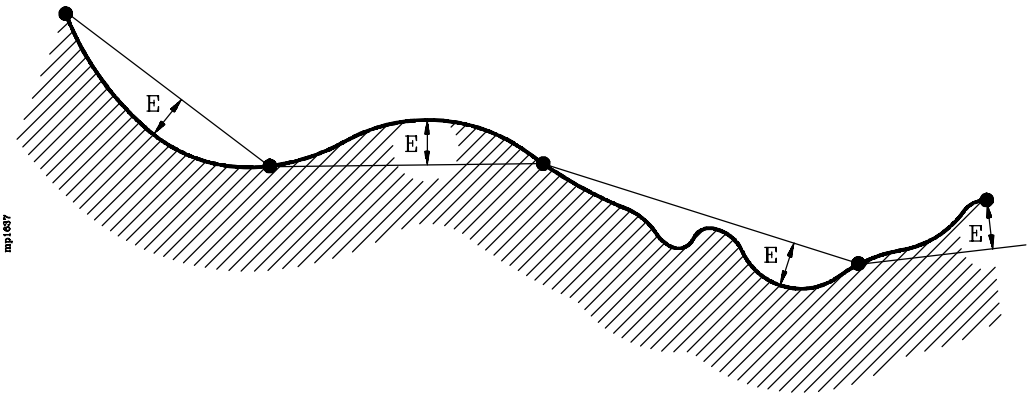


The CNC keeps the probe in constant contact with the surface of the model and it provides the coordinates of a new point after moving, in space and along the programmed path, the distance indicated by parameter "L".

If not programmed or programmed with a value of "0", the canned cycle will assume that the model is **not** to be digitized.

E 5.5 This parameter must be defined when digitizing a part besides tracing it.

It indicates the chordal error or maximum difference allowed between the surface of the model and the segment joining two consecutive digitized points. It is given in the selected work units (millimeters or inches).



If not programmed or programmed with a value of "0", the chordal error will be ignored and a new point will be provided after moving the "L" distance in space and along the programmed path.

G This parameter must be defined when digitizing the model besides tracing it.

Indicates the storing format for the digitized points in the program selected by means of the "**OPEN P**" statement.

G=0 Absolute format.

All points will be programmed in absolute coordinates (G90) and defined by the X, Y and Z axes.

G=1 Absolute filtered format..

All points will be programmed in absolute coordinates (G90); but only those axes whose positions have changed with respect to the previous digitized point will be defined.

G=2 Incremental filtered format..

All points will be programmed in incremental coordinates (G91) and referred to the previous digitized point. Only those axes whose positions have changed with respect to the previous digitized point will be defined.

If not programmed, the canned cycle will assume a value of G0.

F5.5 Defines the sweeping feedrate. It is given in mm/min or inches/min.

BASIC OPERATION:

1.- The probe positions at the point set by parameters X, Y and Z.

2.- The CNC approaches the probe to the model until it touches it.

3.- The probe keeps in constant contact with the surface of the model following it along the programmed path.

If it is to be digitized, (parameters "L" and "E") it will generate a new block per every digitized point in the program previously opened by means of the "**OPEN P**" statement.

4.- Once the canned cycle is finished, the probe will return to the cycle calling point. This move consists of:

* Movement of the probe along the probing axis.
* Movement of the probe in the main work plane.

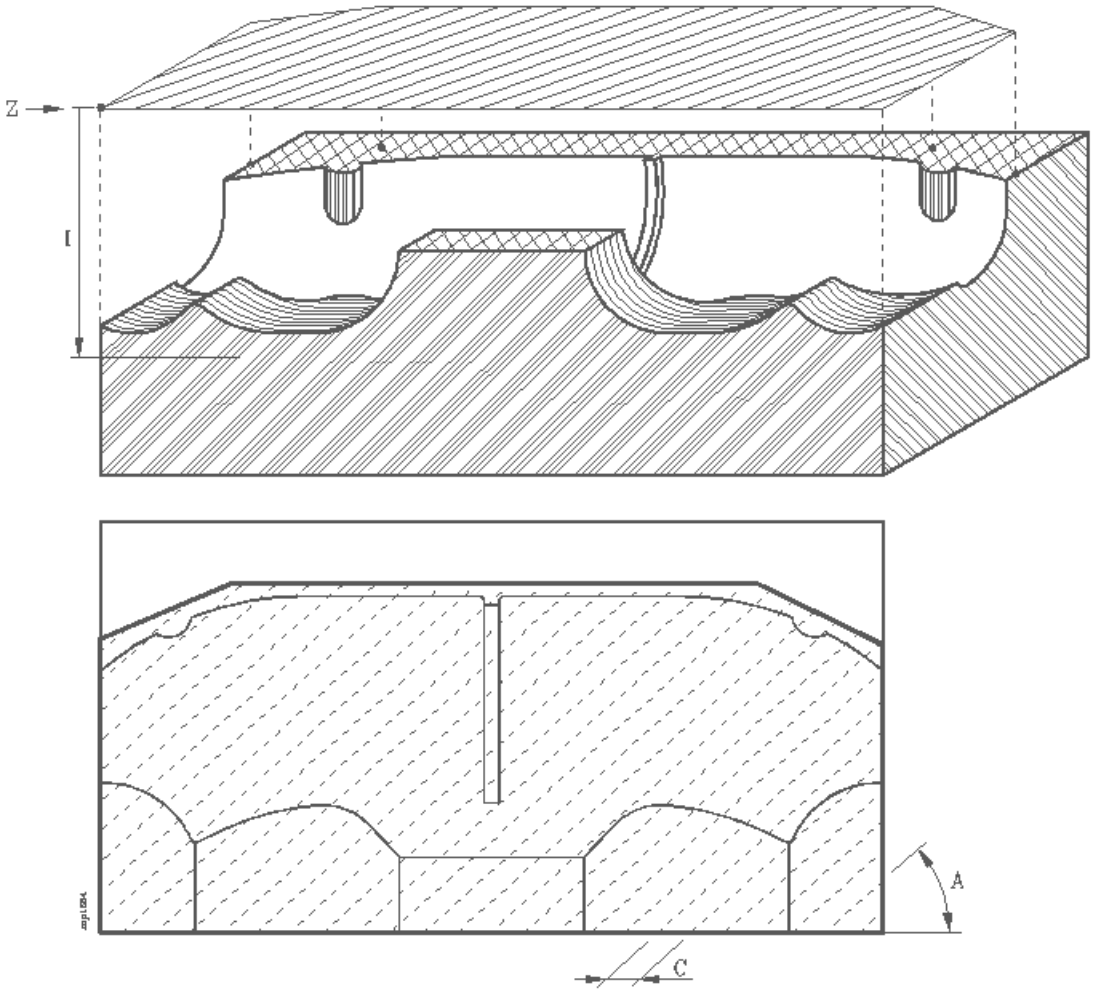
16.7.5 TRACING CANNED CYCLE WITH POLYGONAL SWEEP

With this option it is possible to delimit the tracing area by means of simple geometric elements (straight lines and arcs).

It is also possible to define some zones inside the main tracing area which are not to be traced. These inside zones will be referred to as islands.

The programming format for this cycle is as follows:

(TRACE 5, A, Z, I, C, D, N, L, E, G, H, F, P, U)



A±5.5 Defines the angle of the sweeping paths with respect to the abscissa axis.

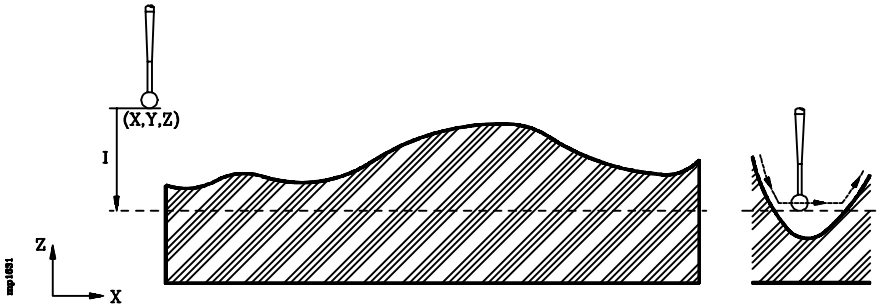
If not programmed, the CNC assumes a value of "A0".

Z±5.5 Absolute theoretical coordinate along the probing axis (longitudinal / perpendicular) where the probe is to be positioned before starting to trace.

It must be off the model and at a safety distance from its outermost surface.

I±5.5 Defines the maximum tracing depth and it will be referred to the coordinate value set by parameter "Z".

If part of the model is out of this area, the tracing will assign this maximum depth to the probing axis and will continue executing the tracing cycle without issuing an error.



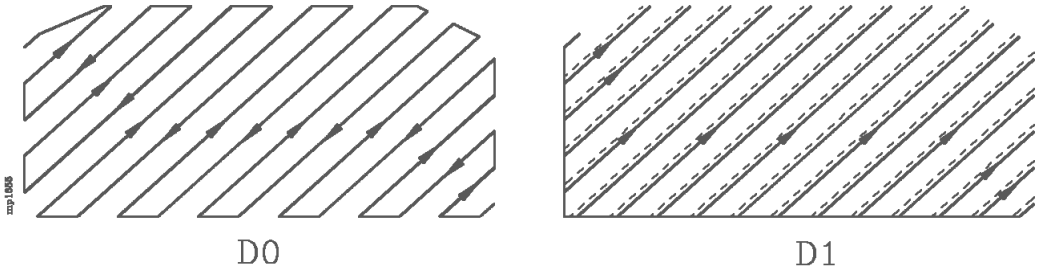
If programmed with a value of "0", the CNC will issue the corresponding error.

C Defines the distance between two consecutive tracing passes.

If programmed with a value of "0", the CNC will issue the corresponding error.

D Indicates how the grid is followed according to the following code:

0 = The tracing is carried out in both directions (zig-zag).
1 = The tracing is carried out following the grid in one direction.



If not programmed, the CNC assumes D0.

N 5.5 Nominal Deflection. Indicates the pressure kept by the probe while sweeping the surface of the model.

The deflection is given in the selected work units (mm or inches) and its value is usually comprised between 0.3mm and 1.5mm.

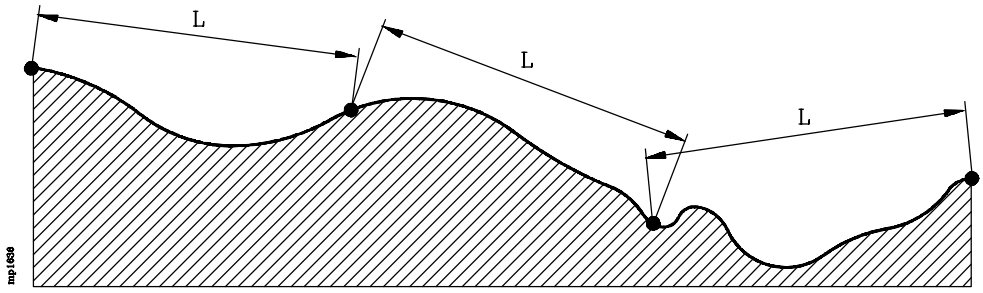
The tracing quality depends upon the amount of deflection being used, the tracing feedrate and the geometry of the model.

In order to prevent the probe from separating from the model, it is advised to use a profile tracing feedrate of about 1000 times the deflection value per minute. For example: for a deflection value of 1mm, the tracing feedrate would be 1m/min.

If not programmed, the canned cycle will assume a value of 1mm (0.03937").

L 5.5 This parameter must be defined when digitizing a part besides tracing it.

It indicates the sweeping step or distance between two consecutive digitized points.

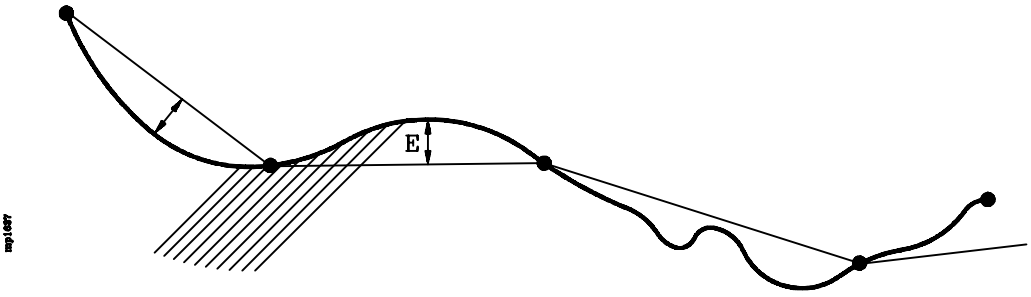


The CNC keeps the probe in constant contact with the surface of the model and it provides the coordinates of a new point after moving, in space and along the programmed path, the distance indicated by parameter "L".

If not programmed or programmed with a value of "0", the canned cycle will assume that the model is **not** to be digitized.

E 5.5 This parameter must be defined when digitizing a part besides tracing it.

It indicates the chordal error or maximum difference allowed between the surface of the model and the segment joining two consecutive digitized points. It is given in the selected work units (millimeters or inches).



If not programmed or programmed with a value of "0", the chordal error will be ignored and a new point will be provided after moving the "L" distance in space and along the programmed path.

G This parameter must be defined when digitizing the model besides tracing it.

Indicates the storing format for the digitized points in the program selected by means of the "**OPEN P**" statement.

G=0 Absolute format.

All points will be programmed in absolute coordinates (G90) and defined by the X, Y and Z axes.

G=1 Absolute filtered format..

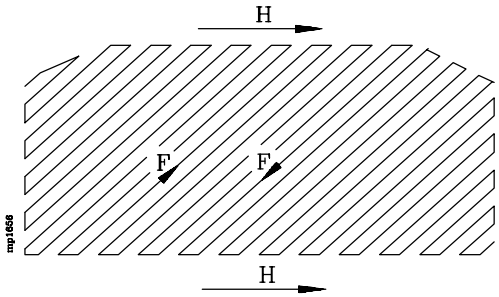
All points will be programmed in absolute coordinates (G90); but only those axes whose positions have changed with respect to the previous digitized point will be defined.

G=2 Incremental filtered format..

All points will be programmed in incremental coordinates (G91) and referred to the previous digitized point. Only those axes whose positions have changed with respect to the previous digitized point will be defined.

If not programmed, the canned cycle will assume a value of G0.

H5.5 Defines the feedrate for the incremental paths. It is given in mm/min or inches/min.



If not programmed, the canned cycle assumes the "F" value (sweeping feedrate).

F5.5 Defines the sweeping feedrate. It is given in mm/min or inches/min.

P (0-9999) Defines the label number of the block where the geometric description of the various profiles of the part starts.

U (0-9999) Defines the label number of the block where the geometric description of the various profiles of the part ends.

All the programmed profiles (outside and islands) must be closed.

The profile programming rules as well as the programming syntax are described later on.

BASIC OPERATION:

1.- The probe positions at the point set by parameters X, Y and Z.

2.- The CNC approaches the probe to the model until it touches it.

3.- The probe keeps in constant contact with the surface of the model following it along the programmed path.

If it is to be digitized, (parameters "L" and "E") it will generate a new block per every digitized point in the program previously opened by means of the "**OPEN P**" statement.

4.- Once the canned cycle is finished, the probe will return to the cycle calling point. This move consists of:

* Movement of the probe along the probing axis.
* Movement of the probe in the main work plane.

16.7.5.1 PROFILE PROGRAMMING RULES

When defining a tracing area and its inside islands or not-tracing zones, the following programming rules must be observed:

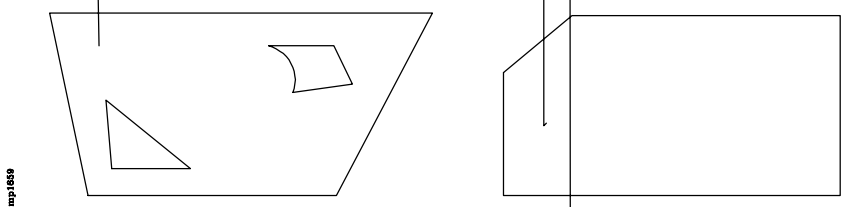
1.- All types of programmed profiles must be closed. The following examples cause a geometry error.

2.- No profile must intersect itself. The following examples cause a geometry error.

3.- The polygon programmed first will be considered by the CNC as the external profile or area to be traced.

All other polygons, if any, must be inside this one and they indicate the islands or inside zones which will not be traced.

4.- It is not required to program inside profiles. Should these be programmed, they must be completely inside the external (main) profile.



5.- An inside profile totally contained within another inside profile cannot be programmed. In this case, only the outermost profile of the two inside ones will be considered.

The CNC verifies all these geometry rules before beginning the execution of the canned cycle adapting the tracing profile to them and displaying the error message when necessary.

16.7.5.2 PROFILE PROGRAMMING SYNTAX

The outside profile as well as the inside ones or islands must be defined by means of simple geometric elements (straight lines and arcs).

The profile programming syntax must observe the following rules:

1.- The block where the geometric description starts must have a label number. This number must be assigned to parameter "P" when defining the canned cycle.

2.- The outside (main) profile or tracing area must be defined first.

No function must be programmed to indicate the end of the profile definition. The CNC considers that the profile has ended when programming function G00 which indicates the beginning of a new profile.

3.- All the inside profiles may be programmed one after another and each one of them must start with function **G00** (which indicates the beginning of a profile).

Warning:

Be sure to program G01, G02 or G03 on the block following the one defining the profile since G00 is modal and the CNC might interpret the following blocks as beginnings of new profiles.

4.- Once the profiles have been defined, assign a label number to the last programmed block. This label number must be assigned to parameter "U" when defining the canned cycle.

5.- The profiles are described as programmed paths and may include the following functions:

G01	Linear interpolation
G02	Clockwise circular interpolation
G03	Counter-clockwise circular interpolation
G06	Absolute arc center coordinates
G08	Arc tangent to previous path
G09	Arc defined by three points
G36	Automatic radius blend (controlled corner rounding)
G39	Chamfer
G53	Programming with respect to machine reference zero (home)
G70	Inch programming
G71	Metric programming
G90	Absolute programming
G91	Incremental programming
G93	Polar origin preset

6.- The profile description does not allow mirror image, scaling factors, pattern rotation, zero offsets, etc.

7.- It is not possible either to program blocks in high level language such as jumps, calls to subroutines or parametric programming.

8.- No other canned cycles can be programmed.

Programming example:

(TRACE 5, A, Z, I, C, D, N, L, E, G, H, F, **P400, U500**)

```
N400 X-260 Y-190 Z4.5                ; Beginning of first outside profile
   G1.....
   .....
   G0 X230 Y170                      ; Beginning of an inside profile
   G1.....
   .....
   G0 X-120 Y90                      ; Beginning of another inside profile
   G2.....
   .....

N500 X-120 Y90                       ; End of geometric description
```

17. COORDINATE TRANSFORMATION

The description of the general coordinate transformation is divided into three basic features:

- Movement in the incline plane (G49)
- Tool movement according to the tool coordinate system (G47)
- TCP transformation, Tool Center Point (G48)

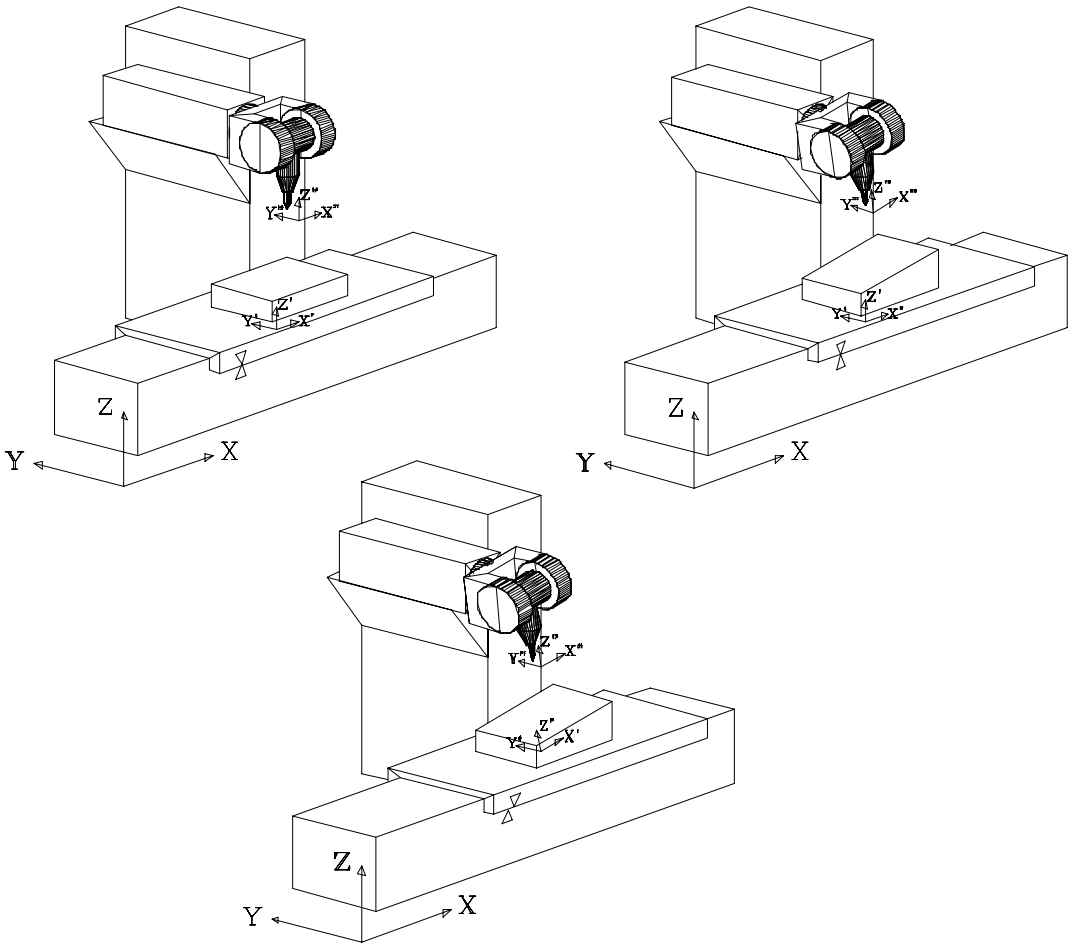
For a better understanding of coordinate transformation, three machine coordinate systems will be considered in the following examples.

- Machine coordinate system. ..... X Y Z in the figures
- Part coordinate system. ..... X' Y' Z' in the figures
- Tool coordinate system. ..... X" Y" Z" in the figures

When no transformation has been done and the spindle is in the starting position, all three types of coordinates coincide. Figure on the left.

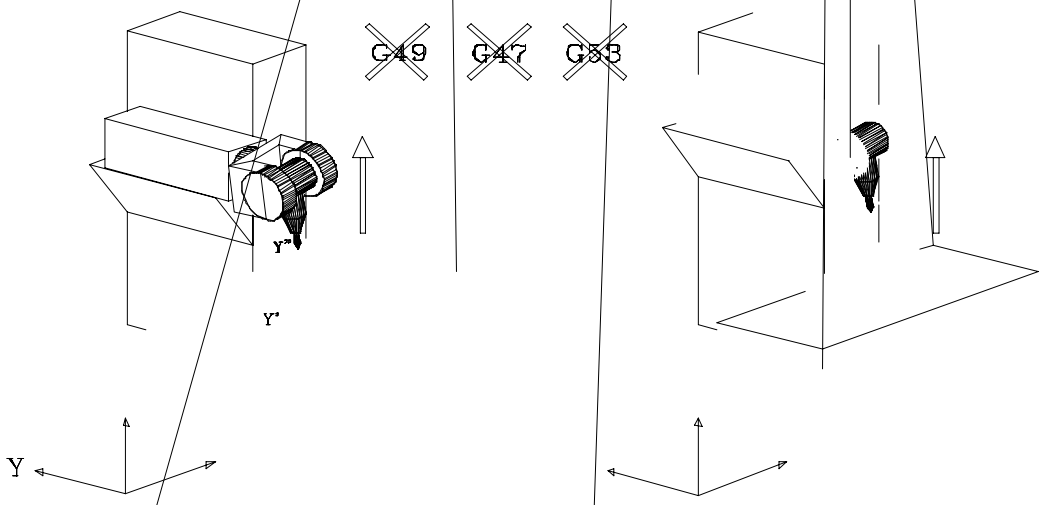
If the spindle turns, the tool coordinate system (X" Y" Z") changes. Figure on the right.

If, also, an incline plane is selected (G49), the part coordinate system also changes (X', Y', Z'). Bottom figure



Case a) ***No transformation has taken place and the spindle is turned.***

If a Z axis movement is programmed (G01 Z), this axis will move according to the part coordinate system which, in this case, coincides with machine coordinates.



Now, to move the tool according to the tool coordinate system, function G47 must be used when programming the movement of the Z axis (G01 G47 Z).

The Z axis will move with respect to the tool coordinates.

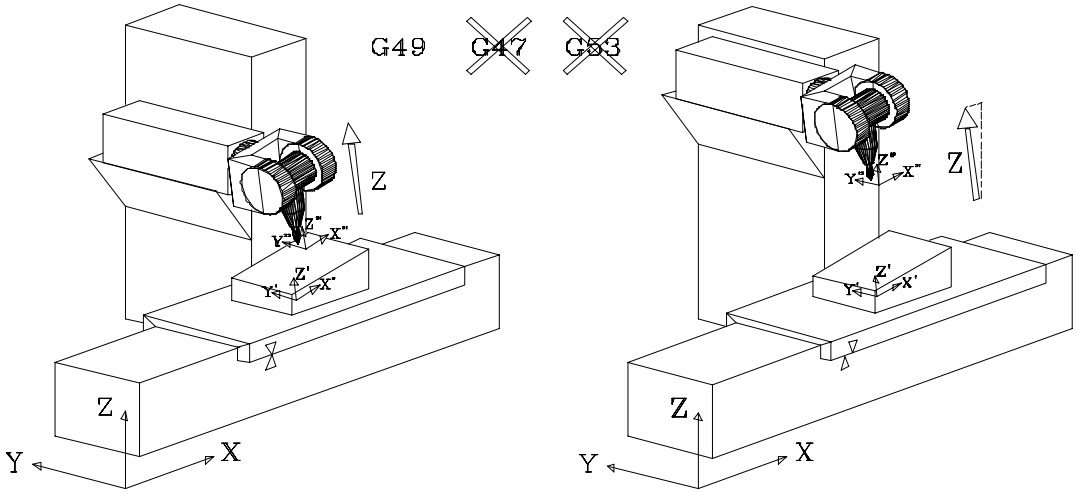
In this type of movements, when the tool coordinate system does not coincide with the machine coordinate system, the CNC moves several axes in order to move the tool according to the part coordinates. In the example, the X and Z axes move.

Function G47 is not modal and only affects the programmed movement.

In order for the jog movements, to be carried out according to the tool coordinate system, the CNC general logic input "TOOLMOVE (M5021" must be activated at the PLC.

Case b) An incline plane has been selected (G49) and the spindle is perpendicular to it

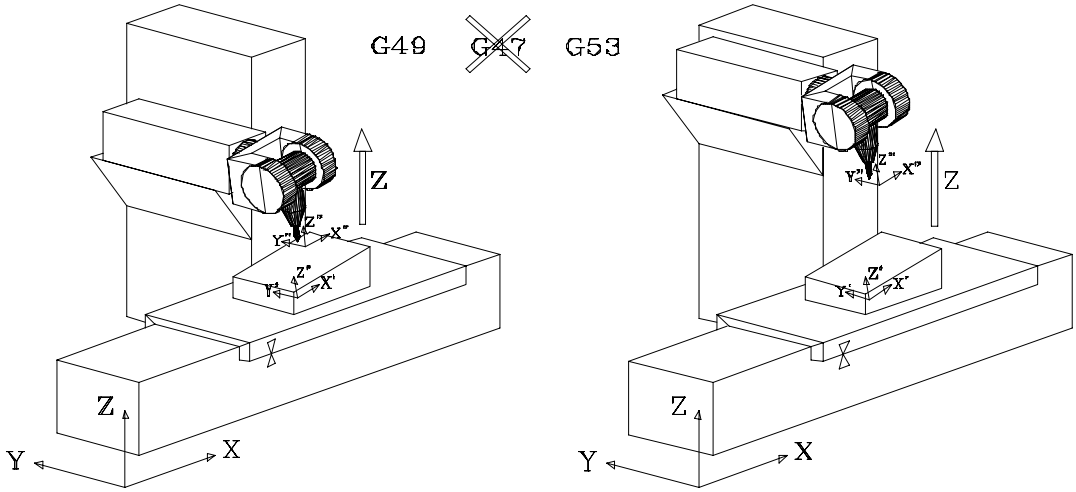
If a Z axis movement is programmed (G01 Z), this axis will move according to the part coordinate system.



In this type of movements, when the part coordinate system does not coincide with the machine coordinate system, the CNC moves several axes in order to move the tool according to the part coordinates. In the example, the X and Z axes move.

To move the tool according to the machine coordinate system, function G53 (programming with respect to home) must be used when programming the movement of the Z axis (G01 G53 Z).

The Z axis will move with respect to home coordinates

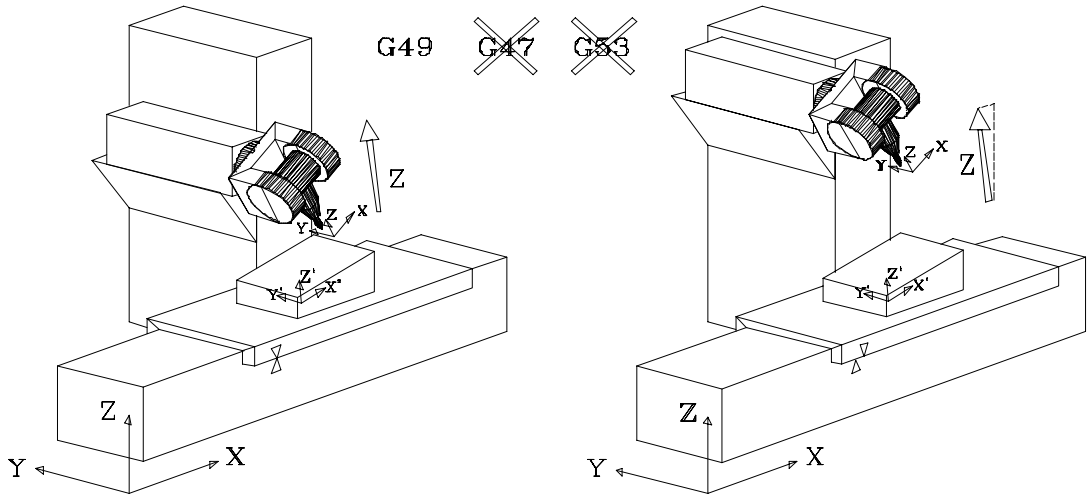


Function G53 is not modal and only affects the programmed movement.

In order for the jog movements to be carried out according to the machine coordinate system, CNC general logic input "MACHMOVE (M5012)" must be activated at the PLC.

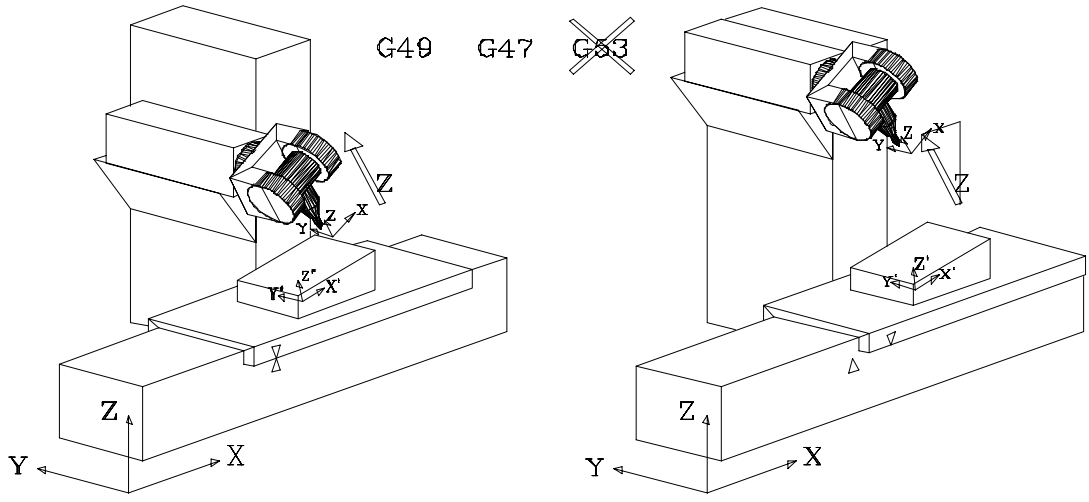
Case c) ***Incline plane selected (G49) and spindle not perpendicular to it***

If a Z axis movement is programmed (G01 Z), this axis will move according to the part coordinate system.



In this type of movements, when the part coordinate system does not coincide with the machine coordinate system, the CNC moves several axes in order to move the tool according to the part coordinates. In the example, the X and Z axes move.

To move the tool according to the tool coordinate system, function G47 must be used when programming the Z axis movement (G01 G47 Z).

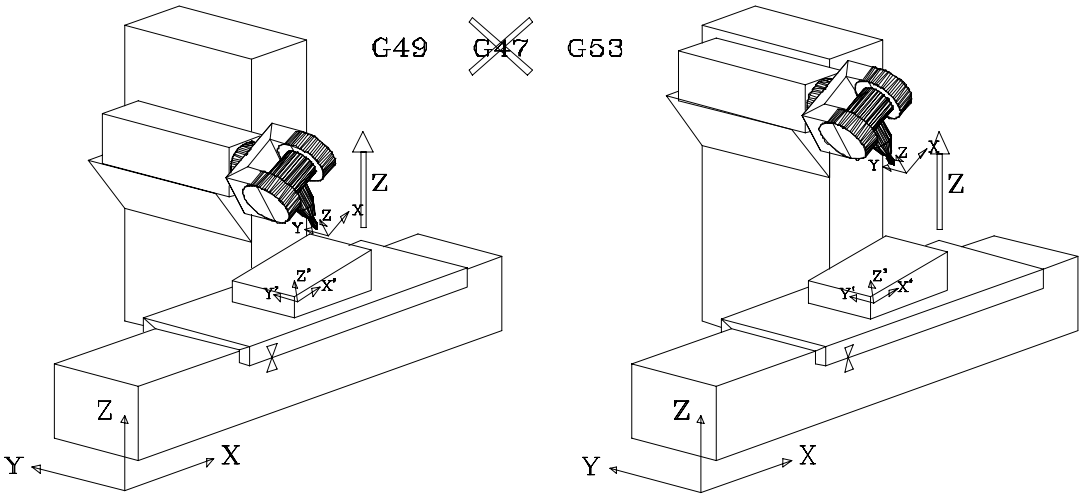


In this type of movements, when the tool coordinate system does not coincide with the machine coordinate system, the CNC moves several axes in order to move the tool according to the part coordinates. In the example, the X and Z axes move.

Function G47 is not modal and only affects the programmed movement.

In order for the jog movements, to be carried out according to the tool coordinate system, the CNC general logic input "TOOLMOVE (M5021" must be activated at the PLC.

To move the tool according to the machine coordinate system, function G53 must be used (programming with respect to home) when programming the Z axis movement (G01 G53 Z).



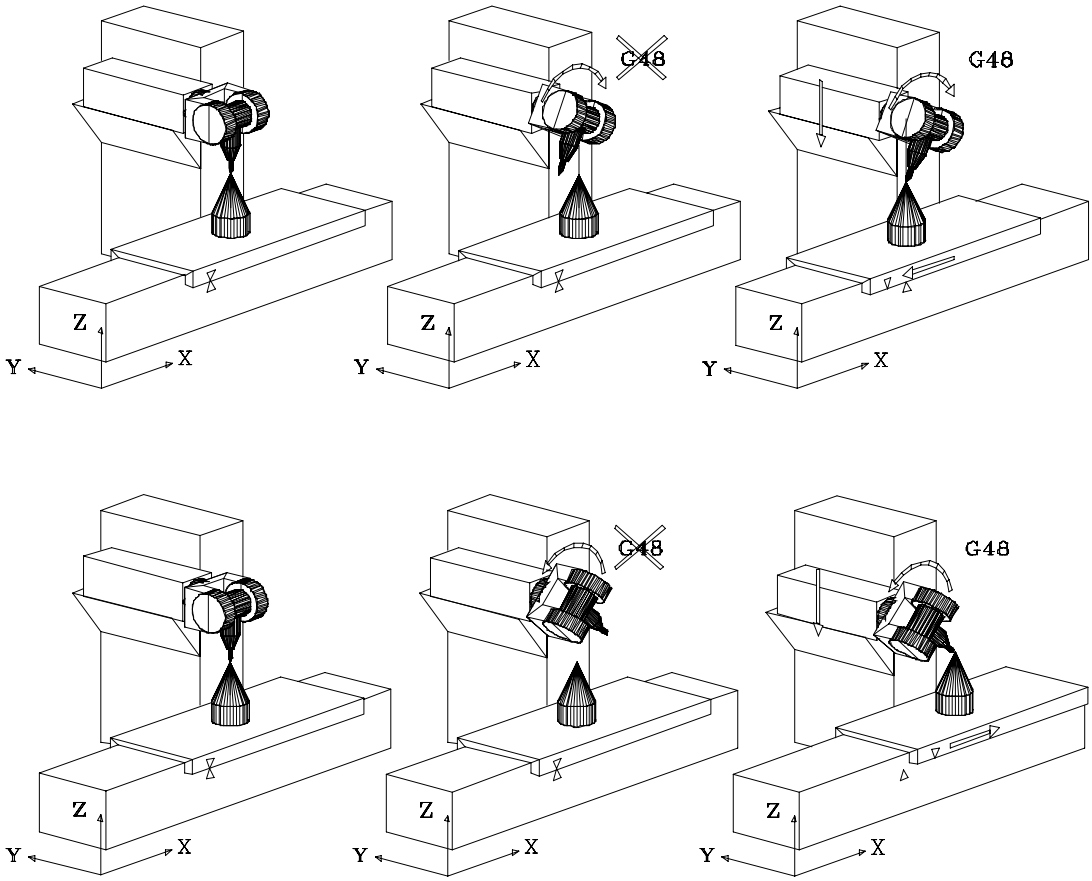
Function G53 is not modal and it only acts in the programmed movement.

In order for the jog movements to be carried out according to the machine coordinate system, CNC general logic input "MACHMOVE (M5012)" must be activated at the PLC.

Case d) ***Working with TCP transformation, Tool Center Point***

When working with TCP transformation, function G48 active, the CNC allows changing the tool orientation without changing the position of its tool tip (part coordinates).

Obviously, the CNC must move several axes of the machine in order to maintain the tool tip position.



Function G48, as described later on, is modal and it indicates when the TCP transformation becomes active and when it is canceled.

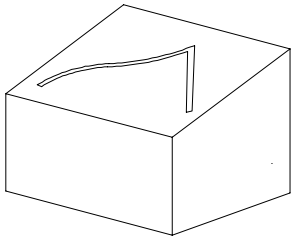
Function G48, TCP transformation, may be used together with function G49 (movement in the Incline Plane) and G47 (movement along the tool axes)

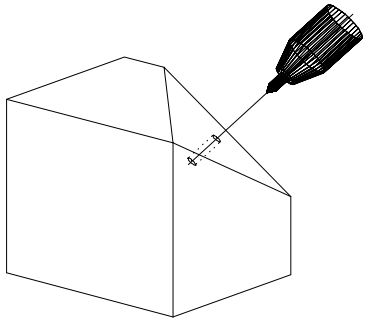
17.1 MOVEMENT IN THE INCLINE PLANE

An incline plane is any plane resulting from a coordinate transformation of the X, Y and Z axes.

With this CNC any plane in space may be selected and any machining performed in it.

The coordinates are programmed as if it were a regular XY plane, but the program will be executed in the indicated incline plane.





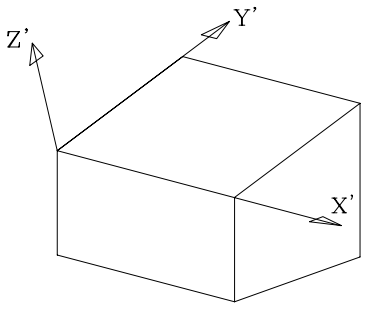
To work with incline planes, always proceed as follows:

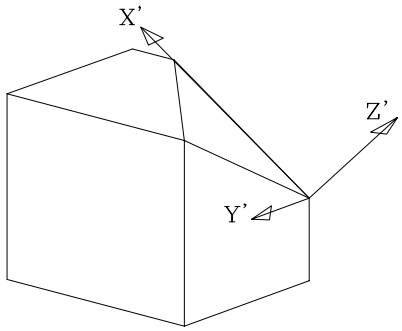
1.- Define, with G49, the incline plane corresponding to the machining operation. G49 is described later on in this chapter.

2.- The CNC variables TOOROF, TOOROS and parameters P297, P298 show the position to be occupied by the spindle rotary axes (main and secondary spindle respectively) in order to orient the tool perpendicular to the indicated incline plane.

3.- To work with the tool perpendicular to the incline plane, rotate the spindle rotary axes to the indicated position.

From this moment on, the X Y axes movements will be carried out along the selected incline plane and the Z axis movement will be perpendicular to it.





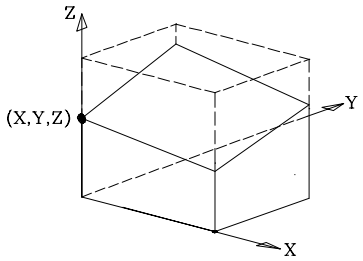
17.1.1 INCLINE PLANE DEFINITION (G49)

Function G49 defines the coordinate transformation or, another words, the incline plane resulting from that transformation.

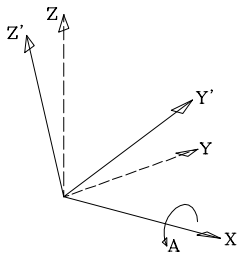
There are several ways to define G49:

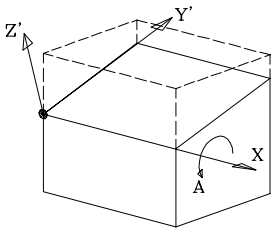
G49 X Y Z A B C Defines the incline plane resulting from rotating around the X axis first and around the Z axis last the amounts indicated in A, B, C respectively.

X, Y, Z Define the coordinate origin of the incline plane. Indicate the X, Y, Z coordinates with respect to the current coordinate origin.

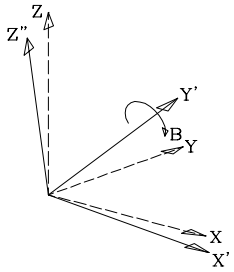


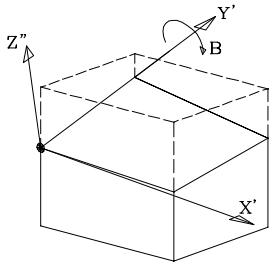
A, B, C Define the incline plane resulting from:
Having rotated around the X axis first the amount indicated by A



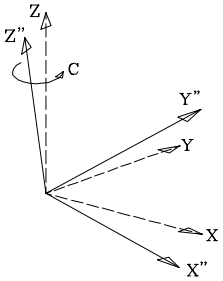


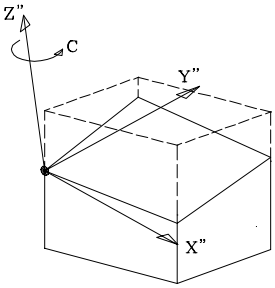
The new coordinate system resulting from this transformation is called X Y' Z' because the Y and Z axes have been rotated.
Then, it must be rotated around the Y' axis the amount indicated by B.





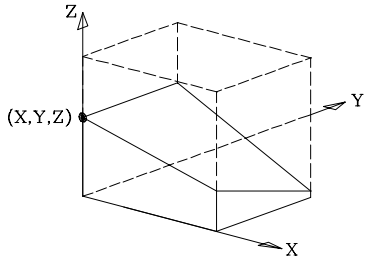
The new coordinate system resulting from this transformation is called X' Y' Z" because the X and Z axes have been rotated.
Finally, rotate around the Z" axis the amount indicated by C.



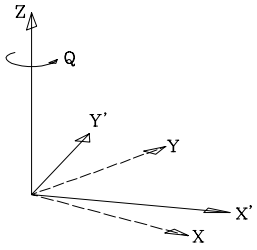


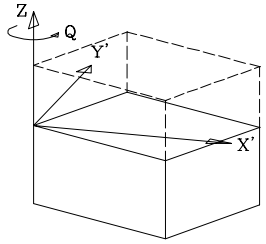
G49 X Y Z Q R S Spheric coordinates. Defines the incline plane resulting from rotating around the Z axis first, then around the Y axis and again around the Z axis the amounts indicated by Q, R, S respectively.

X, Y, Z Define the coordinate origin of the incline plane. They indicate the X, Y, Z coordinates with respect to the current coordinate origin.

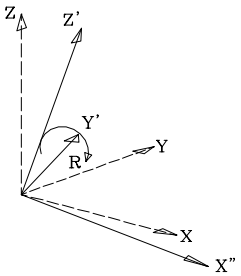


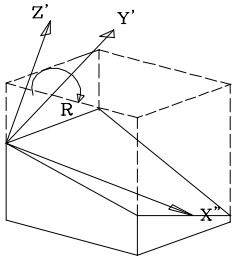
Q, R, S Define the incline plane resulting from:
Having rotated around the Z axis first the amount indicated by Q.



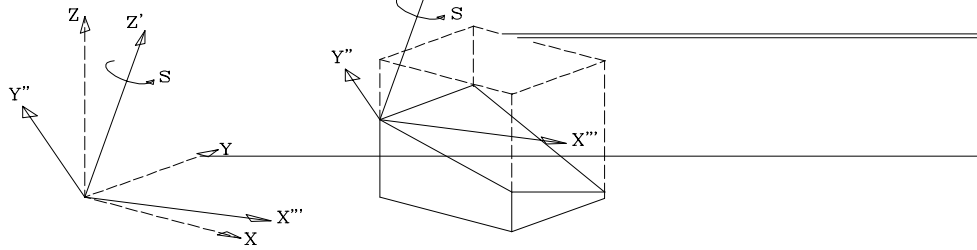


The new coordinate system resulting from this transformation is called X' Y' Z because the X and Y axes have been rotated.
Then, it must be rotated around the Y' axis the amount indicated by R.



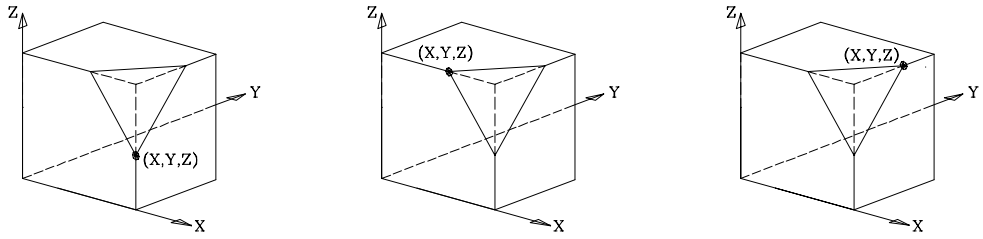


The new coordinate system resulting from this transformation is called X" Y' Z' because the X and Z axes have been rotated
Finally, rotate around the Z' the amount indicated by S

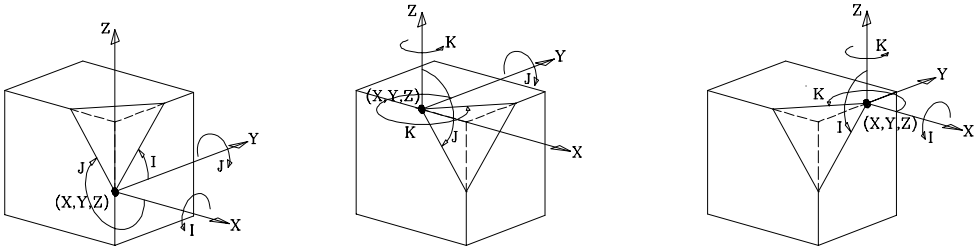


G49 X Y Z I J K R S Defines the incline plane specifying the angles forming the new incline plane with the X Y and Z axes of the machine's coordinate system.

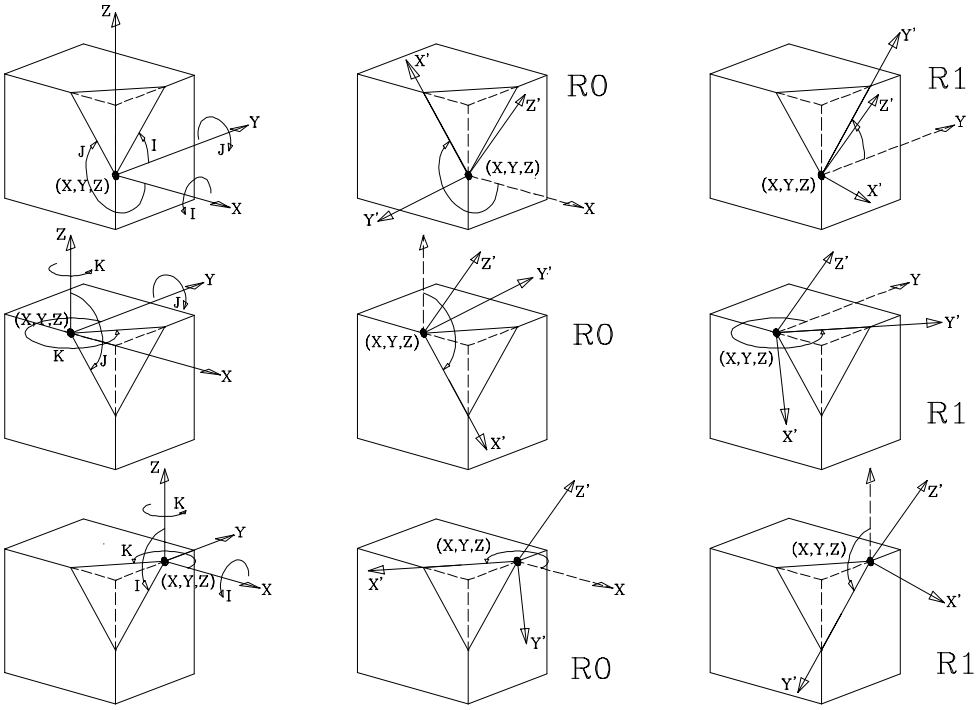
X, Y, Z define the coordinate origin of the incline plane.
Indicate the X, Y, Z coordinates with respect ot the current coordinate origin.



I J K define the angles forming the new incline plane with the X Y and Z axes of the machine's coordinate system. Only two of these three angles are programmed.



R Define which of the axes (X', Y') of the new Cartesian plane is ligned up with the edge. If R0, the X axis is ligned up and If R1, the Y axis is ligned up. If not programmed, a value of R0 is assumed.

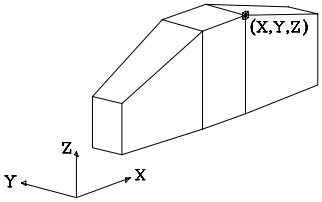


S Lets rotate the coordinates in the new Cartesian plane.

G49 T X Y Z S Defines a new work plane perpendicular to the orientation of the tool.

Is it a good idea to have a swivel or angled spindle (machine parameter "XFORM (P93) set to 2 or 3) when using this type of definition.

T Indicates that one wishes to select a work plane perpendicular to the orientation of the tool.



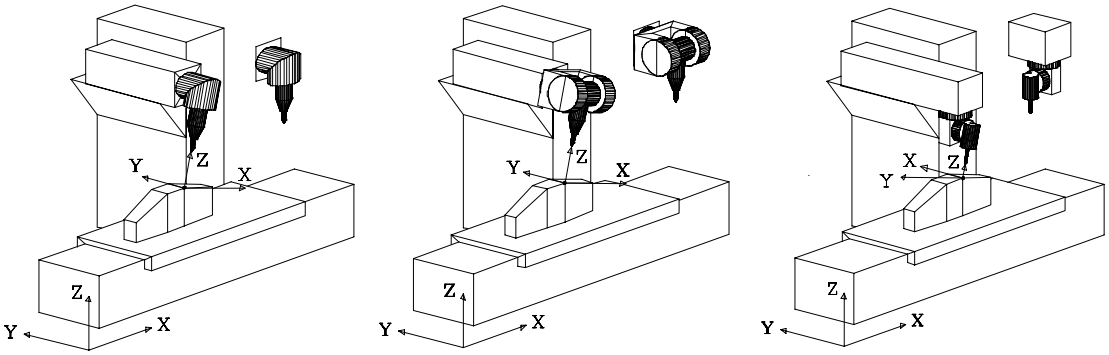
X, Y, Z Define the coordinate origin of the incline plane. Indicate the X, Y, Z coordinates with respect to the current origin.

S Lets rotate the coordinates around the new Z' corresponding to the new work plane.

The new work plane will be perpendicular to the orientation of the tool.
The Z axis keeps the same orientation as the tool.
The orientation of the X, Y axes in the new work plane depends on the spindle type and on how its rotary axes are oriented.

When setting the machine up, it must be set as the spindle's resting position, when the tool is parallel to the Z axis of the machine.

Later, every time the spindle is rotated, the relative tool coordinates will also rotate.



On the two machines on the left, only the main rotary axis has rotated.

But, on the one on the right, both main and secondary rotary axes have rotated in order to achieve the same tool orientation.

On the machine on the right, to orient the X' and Y' axes as in the other 2 cases, one must program:

G49 T XYZ S-90

Programming S-90 means rotating -90º around the new Z' corresponding to the new work plane and, this way, compensate for the rotation of the main rotary axis.

17.1.2 CONSIDERATIONS FOR FUNCTION G49

G49 cannot be programmed in the following instances:
- At the GP model CNC
- From the PLC channel (although it can be programmed from the user channel).
- Within a profile definition for pockets or other cycles.

In order to work with coordinate transformation (G49) the X, Y, Z axes must be defined, form the active trihedron and be linear. The X, Y and Z axes may have GANTRY axes, coupled or synchronized via PLC, associated with them.

When working with coordinate transformation and performing rigid tapping in incline planes, all axes gains (not only for the Z axis) must be adjusted by using the second gains and accelerations.

The parameters associated with G49 are optional. When programming G49 without parameters, the active coordinate transformation is canceled.

G49 is modal and must be programmed alone in the block.

Coordinate transformation is kept active even after turning the CNC off and back on.

To cancel it, G49 must be programmed without parameters. It is also canceled after a home search (G74).

When canceling G49, the CNC recovers the part zero active before G49 was activated.

Zero offsets G54-G59, pattern rotation (G73) and presets (G92, G93) are possible while coordinate transformation is active.

But the following cannot be done:
- Program a new coordinate transformation without previously canceling the previous one.
- Perform tracing operations (G23 through G27).
- Probing (G75).
- Movement against hardstop (G52).

17.1.3 VARIABLES ASSOCIATED WITH FUNCTION G49

Read-only variables associated with the definition of G49:

ORGROX, ORGROY, ORGROZ	New part zero coordinates with respect to home.	
ORGROA, ORGROB, ORGROC	Values assigned to parameters A, B, C	
ORGROI, ORGROJ, ORGROK	Values assigned to parameters I, J, K	
ORGROQ, ORGROR, ORGROS	Values assigned to parameters Q, R, S	
GTRATY	type of G49 programmed	0 = no G49 has been defined
	1 = G49 X Y Z A B C	2 = G49 X Y Z Q R S
	3 = G49 T X Y Z S	4 = G49 X Y Z I J K R S

Every time G49 is programmed, the CNC updates the values of the parameters that have been defined.

For example, when programming G49 XYZ ABC, the CNC
Updates variables ORGROX, Y, Z, A, B, C
The rest of the variables keep their previous values.

Read-Write variables updated by the CNC once G49 has been executed:

When using a swivel or angled spindle, general machine parameter "XFORM (P93)" set to 2 or 3, the CNC shows the following information.

TOOROF Indicates the position to be occupied by the spindle's main rotary axis to orient the tool perpendicular to the indicated incline plane.

TOOROS Indicates the position to be occupied by the spindle's secondary rotary axis to orient the tool perpendicular to the indicated incline plane.

When accessing variable TOOROF or TOOROS the CNC interrupts block preparation and it waits for that command to be executed before resuming block preparation.

17.1.4 PARAMETERS ASSOCIATED WITH FUNCTION G49

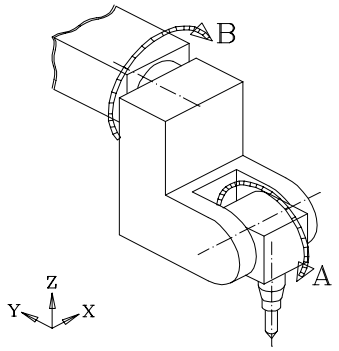
Once G49 has been executed, the CNC updates global parameters P297 and P298:

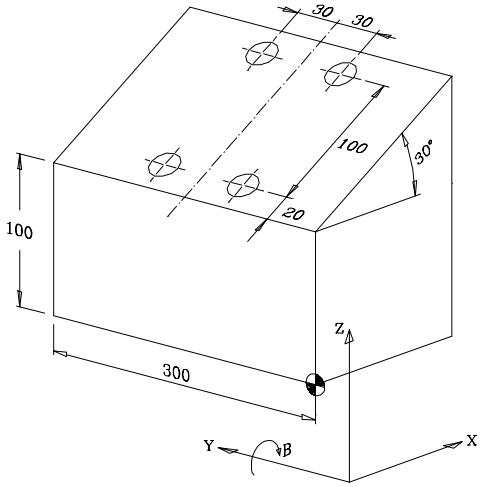
P297 Indicates the position to be occupied by the spindle's main rotary axis to orient the tool perpendicular to the indicated incline plane. It is the same value as shown by the TOOROF variable.

P298 Indicates the position to be occupied by the spindle's secondary rotary axis to orient the tool perpendicular to the indicated incline plane. It is the same value as shown by the TOOROS variable.

Warning: These parameters are global. Therefore, they can be modified by the user or even by probing cycles of the CNC itself.
They should be used after executing G49.
Otherwise, variables TOOROF and TOOROS should be used.

17.1.5 PROGRAMMING EXAMPLE





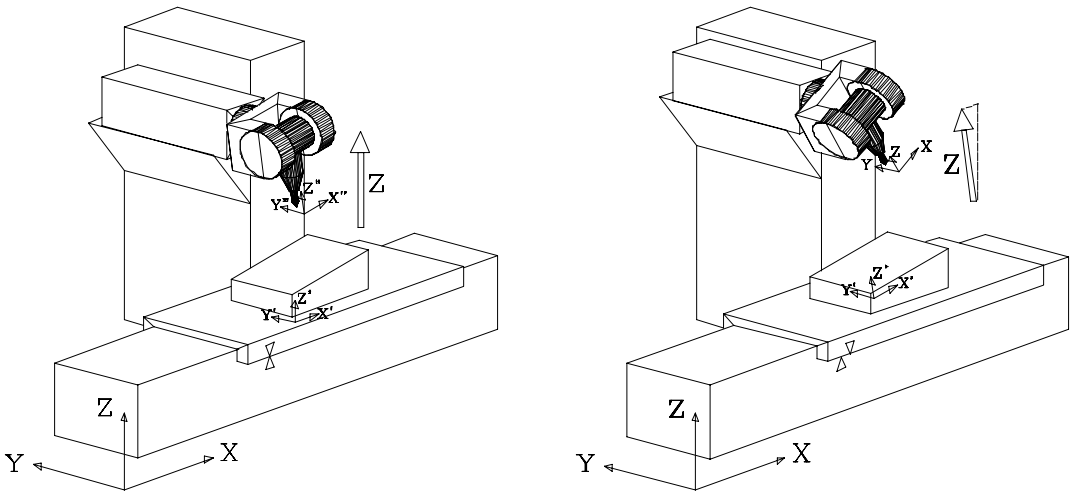
G49 X0 Y0 Z100 B-30	Defines incline plane
G01 AP298 BP297	Orients main axis (B) and secondary axis (A) so the tool is perpendicular to the plane. The programming sequence is ABC, regardless of which one is the main axis or the secondary.
G90 G01 Z5	Tool approach to the work plane.
G90 G01 X20 Y120	Positioning at the 1st point
G??	Machining at the 1st point
G91 G01 Y60	Positioning at the 2nd point
G??	Machining at the 2nd point
G91 G01 X100	Positioning at the 3rd point
G??	Machining at the 3rd point
G91 G01 Y-60	Positioning at the last point
G??	Machining at the last point
G90 G01 Z 20	Withdraw the tool
G49	Cancel incline plane

17.2 MOVEMENT ACCORDING TO THE TOOL COORDINATE SYSTEM (G47)

To move the tool according to the tool coordinate system, function G47 must be used when programming a movement of the Z axis (G01 G47 Z).

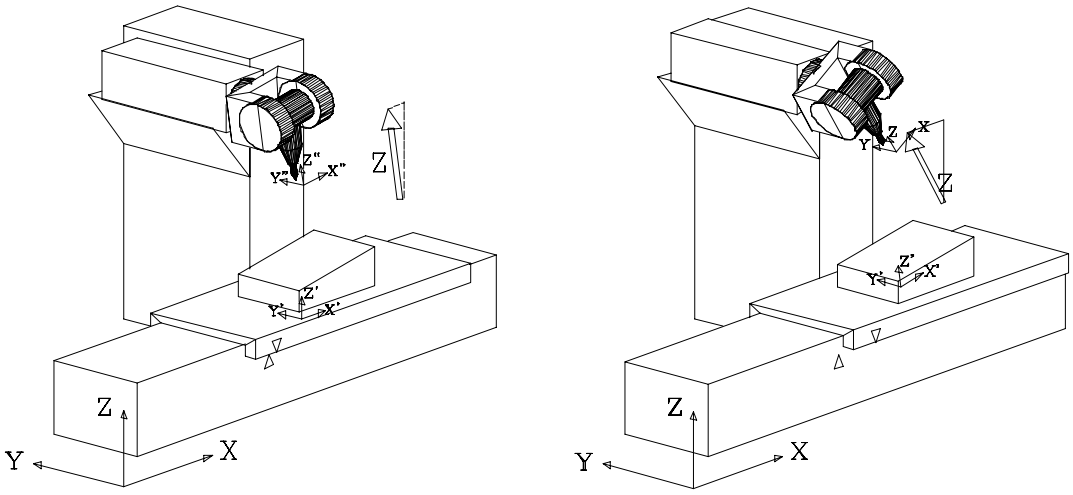
When using this function, a swivel or angled spindle should be utilized (general machine parameter "XFORM (P93)" set to 2 or 3).

When not using this function, the tool moves according to the part coordinate system



In the example on the left, the part coordinates coincide with those of the machine and in the example on the right, an incline plane is active (G49).

To move the tool according to the tool coordinate system, function G47 must be used when programming a movement of the Z axis (G01 G47 Z).



The movements programmed with G47 are always incremental.

Function G47 is not modal and it only acts within the block (linear path) where it has been programmed.

G47 can also be programmed while G48 and G49 are active.

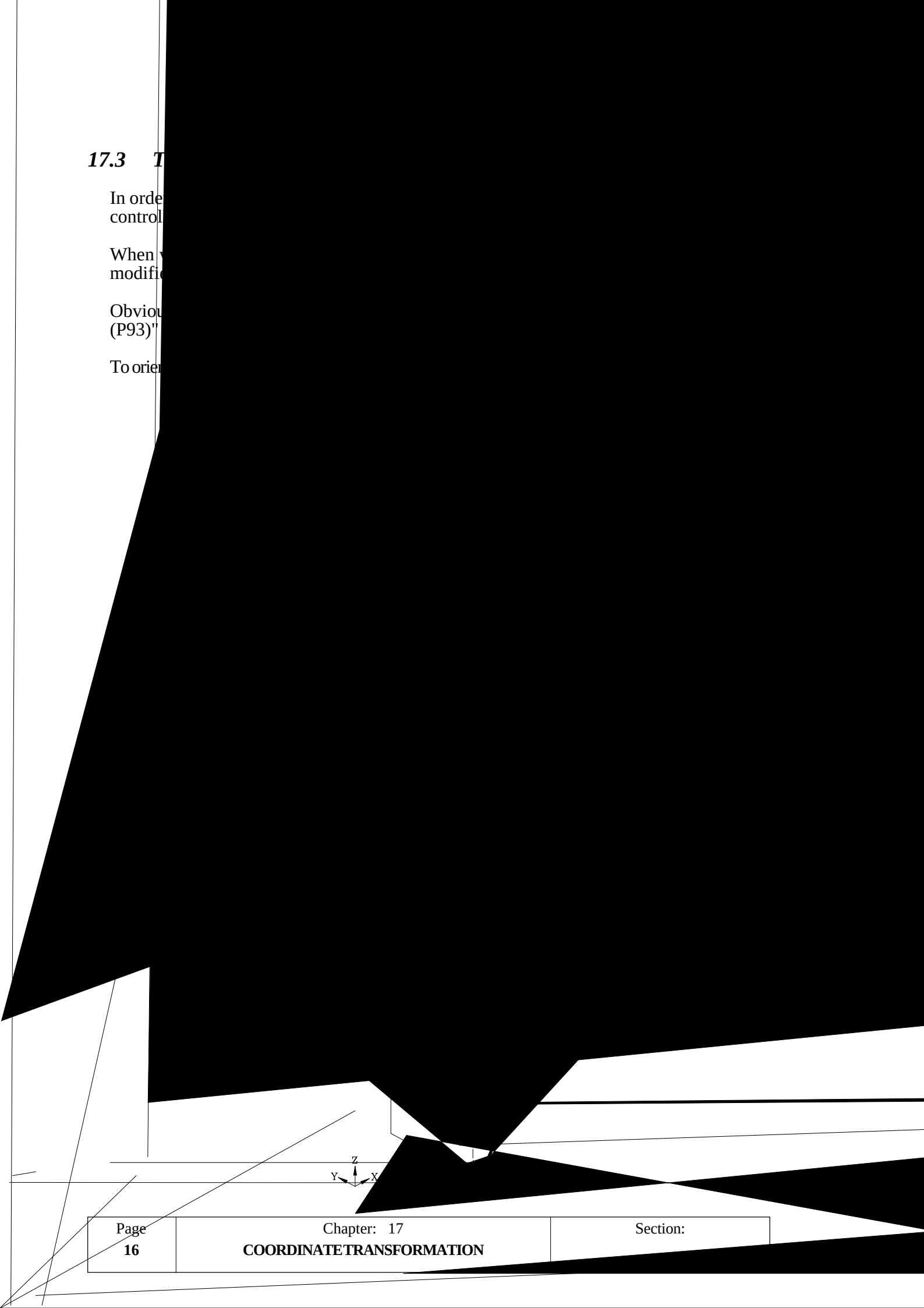
17.3 T

In orde
control

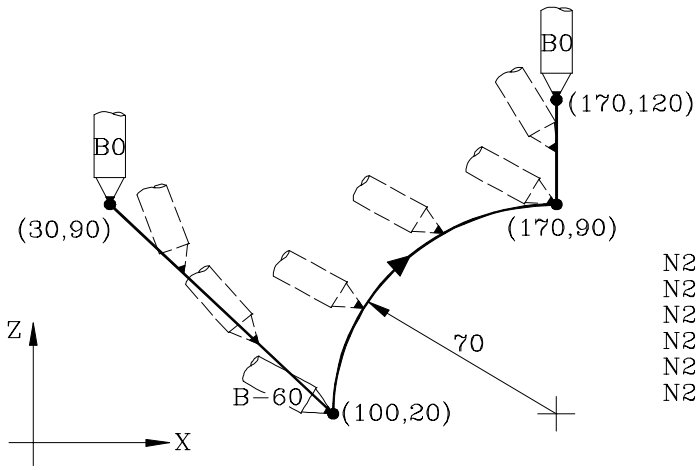
When
modifi

Obviou
(P93)"

To orie



Example a) ***Circular interpolation while maintaining a fixed tool orientation***




```
N20 G18 G90 G01 X30 Z90
N21 G48 S1
N22 G01 X100 Z20 B−60
N23 G03 X170 Z90 I70 K0
N23 G01 X170 Z120 B0
N25 G48 S0
```

Block N20 selects the ZX plane (G18) and positions the tool at the starting point (30,90).

Block N21 turns TCP on.

Block N22 positions the tool at (100,20) orienting it to -60°.

The CNC interpolates the XZB axes executing the programmed linear interpolation while rotating the tool from the starting position (0º) to the programmed final orient position (-60°).

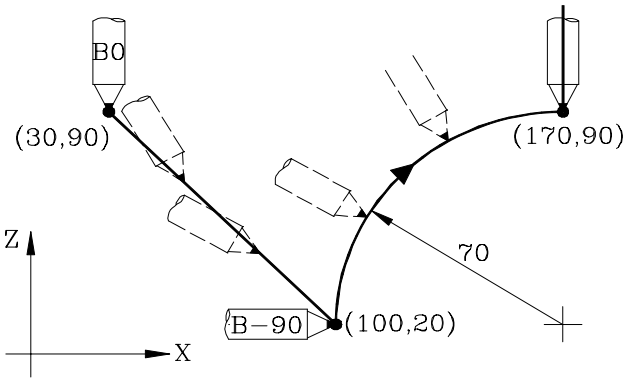
Block N23 does a circular interpolation up to point (170,90) maintaining the same tool orientation for the whole movement.

Block N24 positions the tool at (170,120) orienting it to 0°.

The CNC interpolates the XZB axes executing the programmed linear interpolation while rotating the tool from the current position (-60º) to the programmed final orient position (0°).

Block N25 turns TCP off

Example b) ***Circular interpolation keeping the tool perpendicular to the path***




```
N30 G18 G90 G01 X30 Z90
N31 G4°
N32 G01 X100 Z20 B−90
N33 G03 X170 Z90 I70 K0 B0
N34 G01 X170 Z120
N35 G48 S0
```

Block N30 selects the ZX plane (G18) and positions the tool at the starting point (30,90).

Block N31 turns TCP on.

Block N32 positions the tool at (100,20) orienting it to -90°.

The CNC interpolates the XZB axes executing the programmed linear interpolation while rotating the tool from the starting position (0º) to the programmed final orient position (-90°).

Block N33 defines a circular interpolation up to point (170,90) setting the final tool orientation to (0º).

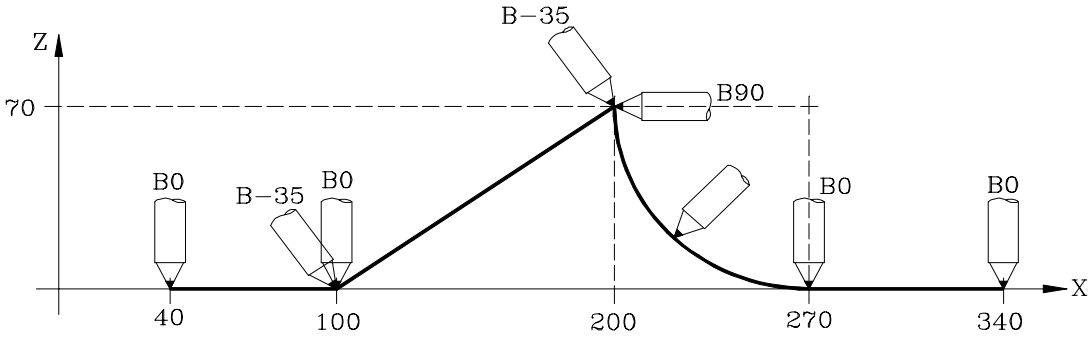
The CNC interpolates the XZB axes executing the programmed circular interpolation while rotating the tool from the current position (-90º) to the programmed final orient position (0°).

Since both orientations are radial, the tool stays radially oriented at all times. In other words, perpendicular to the path.

Block N34 positions the tool at (170,120).

block N35 turns TCP off.

Example c) ***Machining a profile***



G18 G90 ..... selects the ZX plane (G18)
G48 S1 ..... turns TCP on.
G01 X40 Z0 B0 ..... positions the tool at (40,0) orienting it to (0°)
X100 ..... movement to (100,0) with the tool oriented at (0°)
B-35 ..... orients the tool to (-35°)
X200 Z70 ..... movement to (200,70) with the tool orientated to (-35°)
B90 ..... orients the tool to (90°)
G02 X270 Z0 R70 B0 . circular interpolation up to (270,0) keeping the tool perpendicular to the path.
G01 X340 ..... movement to (340,0) with the tool oriented at "0°"
G48 S0 ..... turns TCP on.

17.3.1 CONSIDERATIONS FOR FUNCTION G48

G48 cannot be programmed in the following instances:
At the GP CNC model.
From the PLC channel (although it can be programmed from the user channel).

In order to work with TCP transformation (G48) the X, Y, Z axes must be defined, form the active trihedron and be linear. The X, Y and Z axes may have GANTRY axes, coupled or synchronized via PLC, associated with them.

When working with TCP transformation and performing rigid tapping in incline planes, all axes gains (not only for the Z axis) must be adjusted by using the second gains and accelerations.

TCP transformation is kept active even after turning the CNC off and back on.

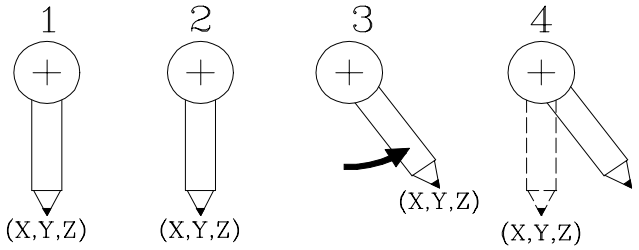
G48 can be programmed while G49 is active and vice versa.

To cancel TCP, program "G48 S0" or G48 without parameters. It is also canceled after a home search (G74).

While TCP is on, it is possible to:
Apply zero offsets G54-G59
Rotate the pattern (coordinate system) (G73)
Preset (G92, G93).
JOG in continuous or incremental moves and by electronic handwheel.

But, it is not possible to:
Perform tracing operations (G23 a G27).
Probe (G75).
Do corner rounding or chamfering because in these instances tool orientation has to be maintained.
Compensate for tool length (G43) because TCP already implies a particular length compensation. CAD/CAM programs usually program the coordinates of the spindle base.

Special care must be taken when turning G48 on and off.
When G48 is on, the CNC shows the coordinates of the tool tip.
When G48 is off, the CNC shows the coordinates of the tool base or theoretical tip (unturned tool).



1.- G48 off. The CNC shows the coordinates of the tool tip.
2.- G48 is turned on. The CNC still shows the coordinates of the tool tip.
3.- The tool is turned. Since G48 is already on, the CNC still shows the coordinates of the tool tip.
4.- G48 is turned off. The CNC shows the coordinates of the theoretical tip (unturned tool).

When working with incline planes and TCP transformation, the following programming order should be used:

G48 S1	Activate TCP transformation
G49	Define the incline plane
G01 AP298 BP297	Position the tool perpendicular to the plane
G	Start the machining operation
.....	Finish the machining operation
G49	Cancel the incline plane
G48	S0 Cancel TCP transformation
M30	End of part program

TCP transformation should be activated first because it lets orient the tool without changing the position of its tip, thus avoiding undesired collisions.

APPENDIX

A. ISO CODE PROGRAMMING 3

B. VARIABLES ASSOCIATED WITH TOOLS 5

C. HIGH LEVEL PROGRAMMING 10

D. KEY CODES 12

E. LOGIC OUTPUTS FOR KEY CODE STATUS 13

F. KEY INHIBITING CODES 14

G. PROGRAMMING ASSISTANCE SYSTEM PAGES 15

H. MAINTENANCE 18

ISO CODE PROGRAMMING

Function	M	D	V	Meaning	Section
G00	*	?	*	Rapid travel	6.1
G01	*	?	*	Linear interpolation	6.2
G02	*		*	Clockwise (helical) circular interpolation	6.3
G03	*		*	Counter-clockwise (helical) circular interpolation	6.3
G04				Dwell/block preparation stop	7.1, 7.2
G05	*	?	*	Round corner	7.3.1
G06			*	Absolute arc center coordinates	6.4
G07	*	?		Square corner	7.3.2
G08			*	Arc tangent to previous path	6.5
G09			*	Arc defined by three points	6.6
G10	*	*		Mirror image cancellation	7.5
G11	*		*	Mirror image on X axis	7.5
G12	*		*	Mirror image on Y axis	7.5
G13	*		*	Mirror image on Z axis	7.5
G14	*		*	Mirror image in the programmed directions	7.5
G15	*		*	Longitudinal axis selection	8.2
G16	*		*	Selection of main plane in two directions	3.2
G17	*	?	*	Main plane X-Y and longitudinal Z	3.2
G18	*	?	*	Main plane Z-X and longitudinal Y	3.2
G19	*		*	Main plane Y-Z and longitudinal X	3.2
G20				Definition of lower work zone limits	3.7.1
G21				Definition of upper work zone limits	3.7.1
G22			*	Activate/cancel work zones	3.7.2
G23			*	Activate tracing	16.3
G24			*	Activate digitizing	16.6
G25				Deactivate tracing/digitizing	16.5
G26			*	Tracing probe calibration	16.2
G27			*	Tracing contour definition	16.4
G28	*		*	Second spindle selection	5..3
G29	*	*		Main spindle selection	5..3
G28-G29			*	Axes toggle	7.9
G32	*		*	Feedrate as an inverted function of time.	6.14
G33	*		*	Threadcutting	6.12
G36			*	Automatic radius blend	6.10
G37			*	Tangential entry	6.8
G38			*	Tangential exit	6.9
G39			*	Automatic chamfer blend	6.11
G40	*	*		Cancellation of tool radius compensation	8.1
G41	*		*	Right-hand tool radius compensation	8.1
G42	*		*	Left-hand tool radius compensation	8.1
G43	*	?	*	Tool length compensation	8.2
G44	*	?	*	Cancellation of tool length compensation	8.2
G45	*		*	Tangential control (G45)	6.15
G47			*	Tool movement according to tool coordinate system	17.2
G48	*		*	TCP transformation	17.3
G49	*		*	Incline plane definition	17.1
G50	*		*	Controlled corner rounding	7.3.3
G51	*		*	Look-Ahead	7.4
G52				Movement until making contact	6.13
G53				Program coordinates with respect to home	4.3
G54	*		*	Absolute zero offset 1	4.4.2
G55	*		*	Absolute zero offset 2	4.4.2
G56	*		*	Absolute zero offset 3	4.4.2
G57	*		*	Absolute zero offset 4	4.4.2
G58	*		*	Additive zero offset 1	4.4.2
G59	*		*	Additive zero offset 2	4.4.2
G60			*	Straight line canned cycle	10.1
G61			*	Rectangular pattern canned cycle	10.2

Function	M	D	V	Meaning	Section
G62			*	Grid pattern canned cycle	10.3
G63			*	Circular pattern canned cycle	10.4
G64			*	Arc pattern canned cycle	10.5
G65			*	Arc-chord pattern canned cycle	10.6
G66			*	Irregular pocket canned cycle	11.1
G67			*	Irregular pocket roughing	11.3
G68			*	Irregular pocket finishing	11.4
G69	*		*	Complex deep hole drilling	9.5.1
G70	*	?	*	Programming in inches	3.3
G71	*	?		programming in millimeters	3.3
G72	*		*	General and specific scaling factor	7.6
G73	*		*	Pattern rotation	7.7
G74			*	Machine reference search	4.2
G75			*	Probing until touching	12.1
G76			*	Probing while touching	12.1
G77	*		*	Slaved axis	7.8.1
G78	*	*		Slaved axis cancellation	7.8.2
G79				Canned cycle parameter modification	9.2.1
G80	*	*		Canned cycle cancellation	9.3
G81	*		*	Drilling cycle	9.5.2
G82	*		*	Drilling cycle with dwell	9.5.3
G83	*		*	Simple deep hole drilling	9.5.4
G84	*		*	Tapping cycle	9.5.5
G85	*		*	Reaming cycle	9.5.6
G86	*		*	Boring cycle with withdrawal in G00	9.5.7
G87	*		*	Rectangular pocket milling cycle	9.5.8
G88	*		*	Circular pocket milling cycle	9.5.9
G89	*		*	Boring cycle with withdrawal in G01	9.5.10
G90	*	?		Programming in absolute	3.4
G91	*	?	*	Programming in incremental	3.4
G92				Coordinate preset/spindle speed limit	4.4.1
G93				Polar origin preset	4.5
G94	*	?		Feedrate in millimeters(inches) per minute	5.2.1
G95	*	?	*	Feedrate in millimeters(inches) per revolution	5.2.2
G96	*		*	Constant cutting point speed	5.4.1
G97	*	*		Constant tool center speed	5.4.2
G98	*	*		Withdrawal to the starting plane	9.5
G99	*		*	Withdrawal to the reference plane	9.5

M means MODAL, i.e., that once programmed, the G function remains active as long as another incompatible G function is not programmed, M02, M30, EMERGENCY, RESET are not programmed or the CNC is not turned on or off.

Letter **D** means BY DEFAULT, i.e., that these will be assumed by the CNC when turned on, after executing M02, M30 or after EMERGENCY or RESET.

In cases indicated with ? it must be interpreted that the DEFAULT of these G functions depends on the settings of the general CNC machine parameters.

V means that the G function is displayed next to the machining conditions in the execution and simulation modes.

INTERNAL CNC VARIABLES

R indicates that the variable can be read.
W indicates that the variable can be modified.

VARIABLES ASSOCIATED WITH TOOLS

Section (13.2.2)

Variable	CNC	PLC	DNC	
TOOL	R	R	R	Number of active tool.
TOD	R	R	R	Number of active tool offset.
NXTOOL	R	R	R	Number of the next requested tool waiting for M06.
NXTOD	R	R	R	

VARIABLES ASSOCIATED WITH FUNCTION G49 *(Section 13.2.4)*

Variables associated with the definition of function G49:

Variable	CNC	PLC	DNC	
ORGROX	R	R	R	X coordinate of the new part zero with respect to home
ORGROY	R	R	R	Y coordinate of the new part zero with respect to home
ORGROZ	R	R	R	Z coordinate of the new part zero with respect to home
ORGROA	R	R	R	Value assigned to parameter A
ORGROB	R	R	R	Value assigned to parameter B
ORGROC	R	R	R	Value assigned to parameter C
ORGROI	R	R	R	Value assigned to parameter I
ORGROJ	R	R	R	Value assigned to parameter J
ORGROK	R	R	R	Value assigned to parameter K
ORGROQ	R	R	R	Value assigned to parameter Q
ORGROR	R	R	R	Value assigned to parameter R
ORGROS	R	R	R	Value assigned to parameter S
GTRATY	R	R	R	Type of G49 programmed (0) no G49 defined, (1) G49 X Y Z A B (2) G49 X Y Z Q R S, (3) G49 T X Y Z S, (4) G49 X Y Z I J K R S

Variables updated by the CNC once G49 has been executed:

TOOROF	R/W	R/W	R/W	Position to be occupied by the spindle's main rotary axis.
TOOROS	R/W	R/W	R/W	Position to be occupied by the spindle's secondary rotary axis.

VARIABLES ASSOCIATED WITH MACHINE PARAMETERS *(Section 13.2.5)*

Variable	CNC	PLC	DNC	
MPGn	R	R	-	Value assigned to general machine parameter (n).
MP(X-C)n	R	R	-	Value assigned to machine parameter (n) of the axis (X-C)
MPSn	R	R	-	Value assigned to machine parameter (n) of the main spindle.
MPSSn	R	R	-	Value assigned to machine parameter (n) of the second spindle.
MPASn	R	R	-	Value assigned to machine parameter (n) of the auxiliary spindle.
MPLCn	R	R	-	Value assigned to machine parameter (n) of the PLC.

VARIABLES ASSOCIATED WITH THE WORK ZONES *(Section 13.2.6)*

Variable	CNC	PLC	DNC	
FZONE	R	R/W	R	Status of work zone 1.
FZLO(X-C)	R	R/W	R	Lower limit of work zone 1 along the selected axis (X/C).
FZUP(X-C)	R	R/W	R	Upper limit of work zone 1 along the selected axis (X/C).
SZONE	R	R/W	R	Status of work zone 2.
SZLO(X-C)	R	R/W	R	Lower limit of work zone 2 along the selected axis (X/C).
SZUP(X-C)	R	R/W	R	Upper limit of work zone 2 along the selected axis (X/C)
TZONE	R	R/W	R	Status of work zone 3.
TZLO(X-C)	R	R/W	R	Lower limit of work zone 3 along the selected axis (X/C).
TZUP(X-C)	R	R/W	R	Upper limit of work zone 3 along the selected axis (X/C).
FOZONE	R	R/W	R	Status of work zone 4.
FOZLO(X-C)	R	R/W	R	Lower limit of work zone 4 along the selected axis (X/C).
FOZUP(X-C)	R	R/W	R	Upper limit of work zone 4 along the selected axis (X/C).

VARIABLES ASSOCIATED WITH FEEDRATES

Section (13.2.7)

Variable	CNC	PLC	DNC	
FREAL	R	R	R	Real feedrate of the CNC in mm/min or inch/min.

Variables associated with function G94

FEED	R	R	R	Active feedrate at the CNC (G94) in mm/min or inch/min.
DNCF	R	R	R/W	Feedrate selected via DNC.
PLCF	R	R/W	R	Feedrate selected via PLC.
PRGF	R	R	R	Feedrate selected by program.

Variables associated with function G95

FPREV	R	R	R	Active feedrate at CNC (G95), in m/rev or inch/rev.
DNCFPR	R	R	R/W	Feedrate selected via DNC.
PLCFPR	R	R/W	R	Feedrate selected via PLC.
PRGFPR	R	R	R	Feedrate selected by program.

Variables associated with function G32

PRGFIN	R	R	R	Feedrate selected by program. In 1/min.

Variables associated with Feedrate Override

FRO	R	R	R	Feedrate Override (%) active at the CNC.
PRGFRO	R/W	R	R	Feedrate Override (%) selected by program.
DNCFRO	R	R	R/W	Feedrate Override (%) selected by DNC.
PLCFRO	R	R/W	R	Feedrate Override (%) selected by PLC.
CNCFRO	R	R	R	Feedrate Override (%) selected from the front panel knob.

VARIABLES ASSOCIATED WITH POSITION VALUES *Section (13.2.8)*

Variable	CNC	PLC	DNC	
PPOS(X-C)	R	-	-	Theoretical programmed position value (coordinate).
POS(X-C)	R	R	R	Real position value of the indicated axis.
TPOS(X-C)	R	R	R	Theoretical (real + lag) position value of the indicated axis.
FLWE(X-C)	R	R	R	Following error of the indicated axis.
DEFLEX	R	R	R	Probe deflection along the X axis.
DEFLEY	R	R	R	Probe deflection along the Y axis.
DEFLEZ	R	R	R	Probe deflection along the Z axis.
DIST(X-C)	R/W	R/W	R	Distance travelled by the indicated axis.
LIMPL(X-C)	R/W	R/W	R	Upper second travel limit.
LIMMI(X-C)	R/W	R/W	R	Lower second travel limit.

VARIABLES ASSOCIATED WITH HANDWHEELS *(Section13.2.9)*

Variable	CNC	PLC	DNC	
HANPF	R	-	-	1st handwheel pulses received since CNC power-up.
HANPS	R	-	-	2nd handwheel pulses received since CNC power-up.
HANPT	R	-	-	3rd handwheel pulses received since CNC power-up.
HANPFO	R	-	-	4th handwheel pulses received since CNC power-up.

VARIABLES ASSOCIATED WITH THE MAIN SPINDLE *(Section 13.2.10)*

Variable	CNC	PLC	DNC	
SREAL	R	R	R	Real spindle speed in r.p.m.
SPEED	R	R	R	Active spindle speed at the CNC.
DNCS	R	R	R/W	Spindle speed selected via DNC.
PLCS	R	R/W	R	Spindle speed selected via PLC.
PRGS	R	R	R	Spindle speed selected by program.
SSO	R	R	R	Spindle Speed Override (%) active at the CNC.
PRGSSO	R/W	R	R	Spindle Speed Override (%) selected by program.
DNCSSO	R	R	R/W	Spindle Speed Override (%) selected via DNC.
PLCSSO	R	R/W	R	Spindle Speed Override (%) selected via PLC.
CNCSSO	R	R	R	Spindle Speed Override (%) selected from front panel.
SLIMIT	R	R	R	Spindle speed limit, in rpm, active at the CNC.
DNCSL	R	R	R/W	Spindle speed limit selected via DNC.
PLCSL	R	R/W	R	Spindle speed limit selected via PLC.
PRGSL	R	R	R	Spindle speed limit selected by program.
POSS	R	R	R	Real Spindle position. Between ±999999999 ten-thousandths º
RPOSS	R	R	R	Real Spindle position. Between 0 and 360º (in ten-thousandths º)
TPOSS	R	R	R	Theoretical Spindle position (real + lag) Between ±999999999 ten-thousandths of a degree.
RTPOSS	R	R	R	Theoretical Spindle position (real + lag). Between 0 and 360º (in ten-thousandths of a degree.
FLWES	R	R	R	spindle following error in Closed Loop (M19) in degrees.

VARIABLES ASSOCIATED WITH THE SECOND SPINDLE *(Section 13.2.11*

Variable	CNC	PLC	DNC	
SSREAL	R	R	R	Real spindle speed in r.p.m.
SSPEED	R	R	R	Active spindle speed at the CNC.
SDNCS	R	R	R/W	Spindle speed selected via DNC.
SPLCS	R	R/W	R	Spindle speed selected via PLC.
SPRGS	R	R	R	Spindle speed selected by program.
SSSO	R	R	R	Spindle Speed Override (%) active at the CNC.
SPRGSO	R/W	R	R	Spindle Speed Override (%) selected by program.
SDNCSO	R	R	R/W	Spindle Speed Override (%) selected via DNC.
SPLCSO	R	R/W	R	Spindle Speed Override (%) selected via PLC.
SCNCSO	R	R	R	Spindle Speed Override (%) selected from front panel.
SSLIMI	R	R	R	Spindle speed limit, in rpm, active at the CNC.
SDNCSL	R	R	R/W	Spindle speed limit selected via DNC.
SPLCSL	R	R/W	R	Spindle speed limit selected via PLC.
SPRGSL	R	R	R	Spindle speed limit selected by program.
SPOSS	R	R	R	Real Spindle position. Between ±999999999 ten-thousandths º
SRPOSS	R	R	R	Real Spindle position. Between 0 and 360º (in ten-thousandths º)
STPOSS	R	R	R	Theoretical Spindle position (real + lag) Between ±999999999 ten-thousandths of a degree.
SRTPOS	R	R	R	Theoretical Spindle position (real + lag). Between 0 and 360º (in ten-thousandths of a degree.
SFLWES	R	R	R	spindle following error in Closed Loop (M19) in degrees.

VARIABLES ASSOCIATED WITH THE PLC *Section (13.2.12)*

Variable	CNC	PLC	DNC	
PLCMSG	R	-	R	Number of the active PLC message with the highest priority.
PLCIn	R/W	-	-	32 PLC inputs starting from (n).
PLCOn	R/W	-	-	32 PLC outputs starting from (n).
PLCMn	R/W	-	-	32 PLC marks starting from (n).
PLCRn	R/W	-	-	Indicated (n) Register.
PLCTn	R/W	-	-	Indicated (n) Timer's count.
PLCCn	R/W	-	-	Indicated (n) Counter's count.

VARIABLES ASSOCIATED WITH GLOBAL AND LOCAL PARAMETERS

(Section 13.2.13)

Variable	CNC	PLC	DNC	
GUP n	-	R/W	-	Global parameter (n) (100-P299).
LUP (a,b)	-	R/W	-	Local parameter (b) and its nesting level (a). (P0-P25).
CALLP	R	-	-	Indicates which local parameters have been defined by means of a PCALL or MCALL instruction (calling a subroutine).

VARIABLES SERCOS

(Section 13.2.14)

Variable	CNC	PLC	DNC	
SETGE(X-C)	W	W	-	Work gear and parameter set for (X-C) axis drive
SETGES	W	W	-	Work gear and parameter set for main spindle drive
SSETGS	W	W	-	Work gear and parameter set for 2nd spindle drive
SVAR(X-C) id	R/W	-	-	Sercos variable for (X-C) axis identifier "id"
SVARS id	R/W	-	-	Sercos variable for main spindle identifier "id"
SSVAR id	R/W	-	-	Sercos variable for 2nd spindle identifier "id"
TSVAR(X-C) id	R	-	-	Third attribute of the sercos variable of (X-C) axis identifier "id"
TSVARS id	R	-	-	Third attribute of the sercos variable of main spindle identifier "id"
TSSVAR id	R	-	-	Third attribute of the sercos variable of 2nd spindle identifier "id"

OTHER VARIABLES

(Section 13.2.15)

Variable	CNC	PLC	DNC	
OPMODE	R	R	R	Operating mode.
OPMODA	R	R	R	Operating mode when working in the main channel.
OPMODB	R	R	R	Type of simulation.
OPMODC	R	R	R	Axes selected by handwheel.
NBTOOL	R	R	R	Number of the tool being managed
PRGN	R	R	R	Number of the program in execution.
BLKN	R	R	R	Label number of the last executed block.
GSn	R	-	-	Status of the indicated G function (n).
GGSA	-	R	R	Status of functions G00 thru G24.
GGSB	-	R	R	Status of functions G25 thru G49.
GGSC	-	R	R	Status of functions G50 thru G74.
GGSD	-	R	R	Status of functions G75 thru G99.
MSn	R	-	-	Status of the indicated M function (n)
GMS	-	-	R	Status of M functions: M (0..6, 8, 9, 19, 30, 41..44)
PLANE	R	R	R	Axes which form the active main plane.
LONGAX	R	R	R	Axis affected by the tool length compensation (G15).
MIRROR	R	R	R	Active mirror images.
SCALE	R	R	R	Active general Scaling factor.
SCALE(X-C)	R	R	R	Scaling Factor applied only to the indicated axis.
ORGROT	R	R	R	Rotation angle (G73) of the coordinate system in degrees.
ROTPF	R	-	-	Abscissa of rotation center.
ROTPS	R	-	-	Ordinate of rotation center.
PRBST	R	R	R	Returns probe status.
CLOCK	R	R	R	System clock in seconds.
TIME	R	R	R/W	Time in Hours, minutes and seconds.
DATE	R	R	R/W	Date in Year-Month-Day format
TIMER	R/W	R/W	R/W	Clock activated by PLC, in seconds.
CYTIME	R	R	R	Time to execute a part in hundredths of a second.
PARTC	R/W	R/W	R/W	Part counter of the CNC.
FIRST	R	R	R	Flag to indicate first time of program execution.
KEY	R/W*	R/W	R/W	keystroke code.
KEYSRC	R/W	R/W	R/W	Keystroke source, 0=keyboard, 1=PLC, 2=DNC
ANAIn	R	R	R	Voltage (in volts) of the indicated analog input (n).
ANAOn	W	W	W	Voltage (in volts) to apply to the indicated output (n).
CNCERR	-	R	R	Active CNC error number.
PLCERR	-	-	R	Active PLC error number.
DNCERR	-	R	-	Number of the error generated during DNC communications.
AXICOM	R	R	R	Pair of axes toggled with function G28
TANGAN	R	R	R	Associated with G45. Angular position, in degrees, with respect to programmed path.

Warning: The "KEY" variable can be "written" at the CNC only via the user channel.

HIGH LEVEL PROGRAMMING

DISPLAY STATEMENTS *(Section 14.2)*

(ERROR whole number, "error text")	Stops execution of program and displays indicated error.
(MSG "message")	Displays indicated message.
(DGWZ expression 1, ... expression 6)	Define the graphics display area

ENABLING/DISABLING STATEMENTS *(Section 14.3)*

(ESBLK and DSBLK)	The CNC executes all the blocks which are found between ESBLK and DSBLK as if they were a single block.
(ESTOP and DSTOP)	Enable (ESTOP) and disable (DSTOP) of the Stop key and the external Stop signal (PLC)
(EFHOLD and DFHOLD)	Enable (EFHOLD) and disable (DFHOLD) of the Feed-Hold input (PLC)

FLOW CONTROLLING STATEMENTS *(Section 14.4)*

(GOTO N(expression))	Causes a jump within the same program, to the block defined by label N(expression)
(RPT N(expression), N(expression))	Repeats the execution of the part of a program existing between two blocks defined by means of labels N(expression)
(IF condition <action1> ELSE <action2>)	Analyzes the given condition which must be a relational expression. If the condition is true (result equals 1), <action1> will be executed, otherwise (result equals 0) <action2> will be executed.

SUBROUTINE STATEMENTS *(Section 14.5)*

(SUB integer)	Definition of subroutine
(RET)	End of subroutine
(CALL (expression))	Call to subroutine
(PCALL (expression, (assignment statement), (assignment statement),...)	Call to a subroutine. Besides, allows the initialization, by means of assignment statements, of up to 26 local parameters of this subroutine.
(MCALL (expression), (assignment statement), (assignment statement),...)	The same as PCALL, but converting the subroutine indicated into a modal subroutine.
(MDOFF)	Cancellation of modal subroutine
(PROBE (expression),(assignment statement), (assignment statement),...)	Executes a probing canned cycle, its parameters being initialized by means of assignment statements.
(DIGIT (expression),(assignment statement), (assignment statement),...)	Executes a digitizing canned cycle, its parameters being initialized by means of assignment statements.
(TRACE (expression),(assignment statement), (assignment statement),...)	Executes a tracing canned cycle, its parameters being initialized by means of assignment statements.
(REPOS X, Y, Z, ...)	It must always be used inside interruption subroutines and it facilitates the repositioning of the machine axes to the interruption point.

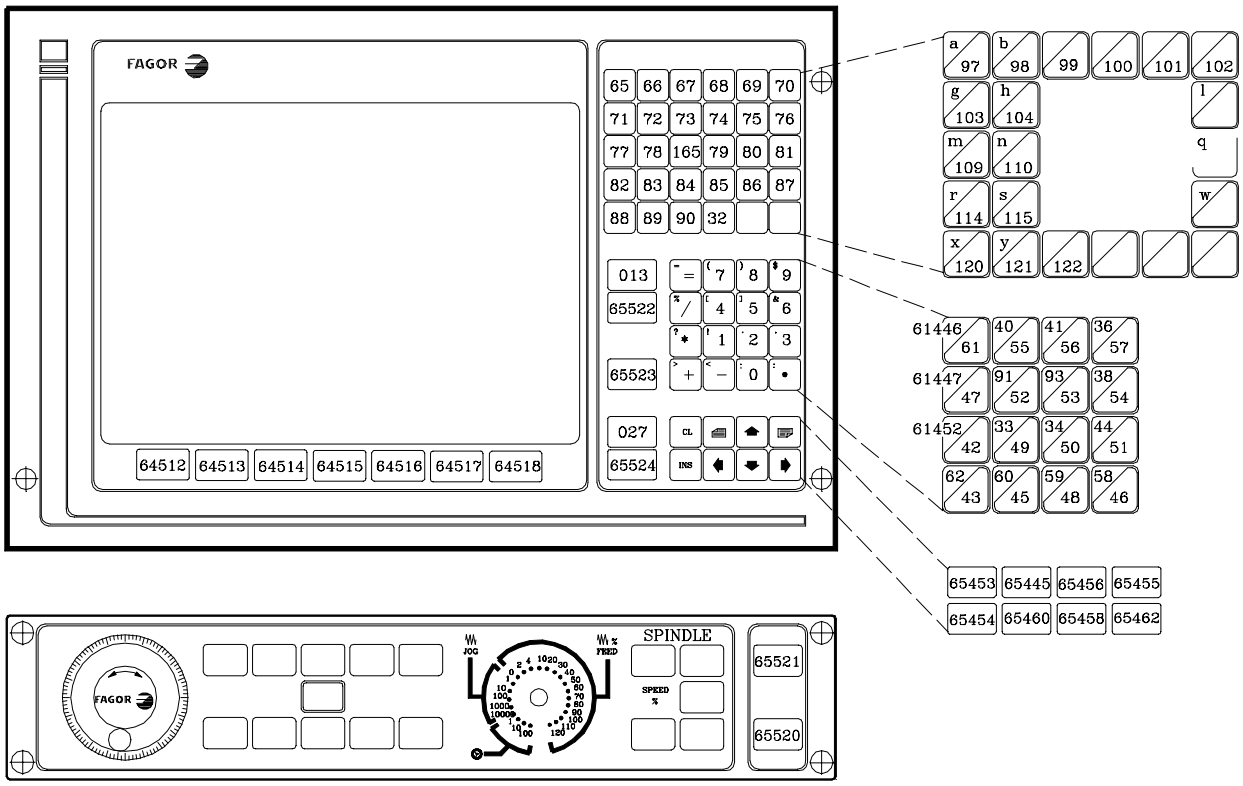
PROGRAM STATEMENTS *(Section 14.6)*

(EXECP(expression), (directory) Starts the execution of the program	
(OPEN P(expression), (destination directory), A/D, "program comment")	Starts generating a new program and allows it to be associated with a program comment.
(WRITE <block text>)	Adds the information contained in <block text> after the last program block of the program which was being generated with OPEN P, as a new program block.

CUSTOMIZING STATEMENTS *(Section 14.7)*

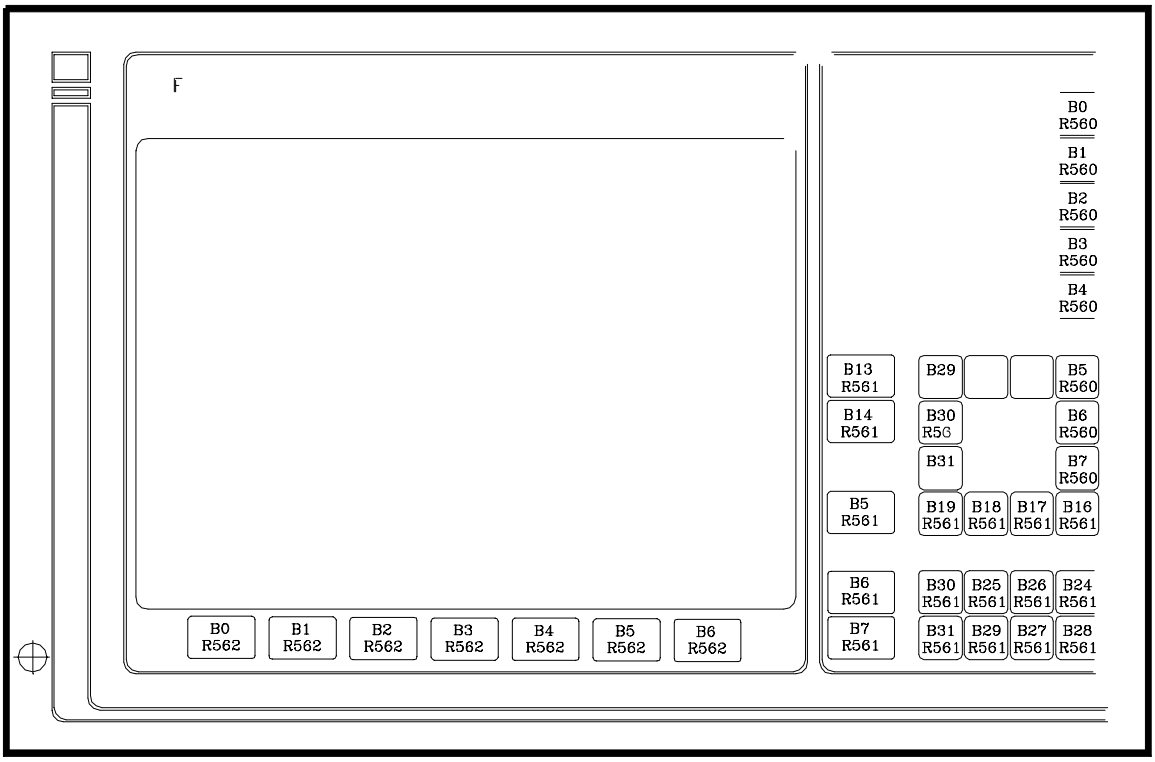
(PAGE(expression))	Displays the user page number (0-255) or system page number (>1000) indicated.
(SYMBOL (expression 1),(expression 2),(expression 3)	Displays the symbol (0-255) indicated by expression 1 Its position on the screen is defined by expression 2 (row,0-639) and by expression 3(column,0-335).
(IB(expression)=INPUT"text",format)	Displays the text indicated in the data input window and stores the data input by the user in the input variable (IBn).
(ODW(expression 1), (expression 2), (expression 3)	Defines and draws a white window on screen (1 row x 14 columns). Its position on screen is defined by expression 2(row) and by expression 3 (column).
(DW (expression 1)=(expression 2), DW(expression 3) = (expression 4),...)	Displays the numerical data indicated by expression 2,4,.. in windows indicated by the value of expression 1,3....
(SK (expression 1)="text 1", (expression 2)="text 2",...)	Defines and displays the new softkey menu indicated.
(WKEY)	Stops the execution of a program until a key is pressed.
(WBUF"text"(expression))	Adds the text and the value of the expression, once this has been evaluated, to the block which is being edited and in the data input window.
(SYSTEM)	Ends the execution of user customized program and returns to standard CNC menu.

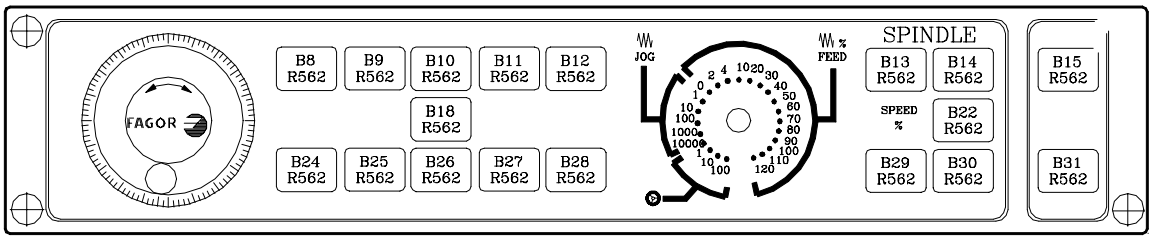
KEY CODES

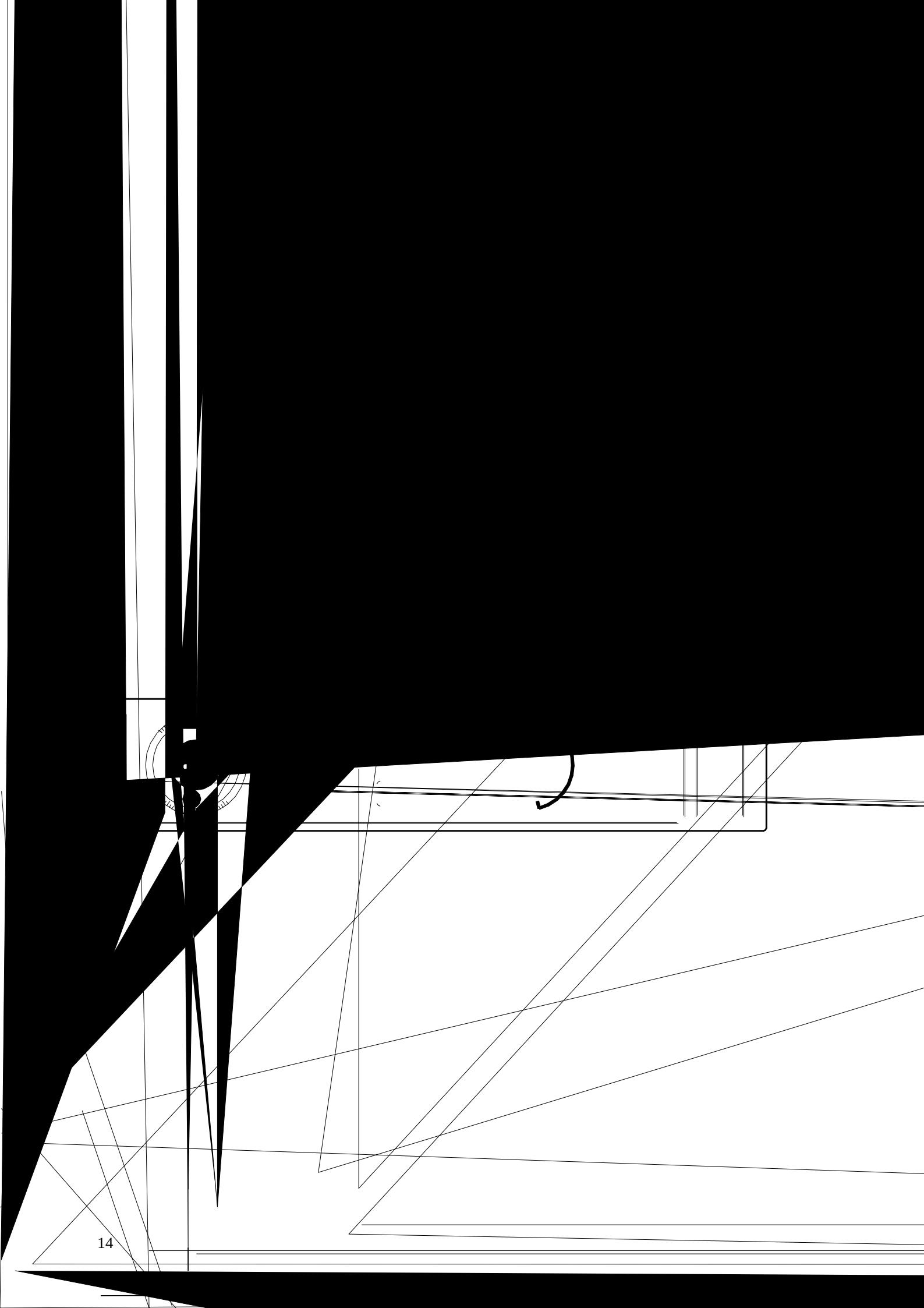

FAGOR
65 66 67 68 69 70
71 72 73 74 75 76
77 78 165 79 80 81
82 83 84 85 86 87
88 89 90 32
013
65522
65523
027
65524
64512 64513 64514 64515 64516 64517 64518
a 97
b 98
99
100
101
102
g 103
h 104
l
m 109
n 110
q
r 114
s 115
w
x 120
y 121
122
61446 61 40 55 41 56 36 57
61447 47 91 52 93 53 38 54
61452 42 33 49 34 50 44 51
62 43 60 45 59 48 58 46
65453 65445 65456 65455
65454 65460 65458 65462

FAGOR
JOG
FEED
SPINDLE
SPEED %
65521
65520

LOGIC OUTPUTS FOR KEY CODE STATUS







PROGRAMMING ASSISTANCE SYSTEM PAGES

These pages can be displayed by means of the high level mnemonic “PAGE”. They all belong to the CNC system and are used as help pages for their respective functions.

GLOSSARY HELP

Page 1000 Preparatory functions G00-G09.
Page 1001 Preparatory functions G10-G19.
Page 1002 Preparatory functions G20-G44.
Page 1003 Preparatory functions G53-G59.
Page 1004 Preparatory functions G60-G69.
Page 1005 Preparatory functions G70-079.
Page 1006 Preparatory functions G80-G89.
Page 1007 Preparatory functions G90-G99.
Page 1008 Miscellaneous (auxiliary) functions M.
Page 1009 Miscellaneous M functions with the symbol for next page.
Page 1010 Coincides with 250 of the directory if it exists.
Page 1011 Coincides with 251 of the directory if it exists.
Page 1012 Coincides with 252 of the directory if it exists.
Page 1013 Coincides with 253 of the directory if it exists.
Page 1014 Coincides with 254 of the directory if it exists.
Page 1015 Coincides with 255 of the directory if it exists.
Page 1016 High level language listing (from A to G)
Page 1017 High level language listing (from H to N)
Page 1018 High level language listing (from 0 to S)
Page 1019 High level language listing (from T to Z)
Page 1020 High level accessible variables (1st part)
Page 1021 High level accessible variables (2nd part)
Page 1022 High level accessible variables (3rd part)
Page 1023 High level accessible variables (4th part)
Page 1024 High level accessible variables (5th part)
Page 1025 High level accessible variables (6th part)
Page 1026 High level accessible variables (7th part)
Page 1027 High level accessible variables (8th part)
Page 1028 High level accessible variables (9th part)
Page 1029 High level accessible variables (10th part)
Page 1030 High level accessible variables (11th part).
Page 1031 High level accessible variables (12th part).
Page 1032 Arithmetic operators.

SYNTAX ASSISTANCE: ISO LANGUAGE

Page 1033 Program block structure
Page 1034 Positioning and linear interpolation: G00,G01 (1st part)
Page 1035 Positioning and linear interpolation: G00,G01 (2nd part)
Page 1036 Circular-helical interpolation: G02, G03 (1st part)
Page 1037 Circular-helical interpolation: G02, G03 (2nd part)
Page 1038 Circular-helical interpolation: G02, G03
Page 1039 Arc tangent to previous path: G08 (1st part)
Page 1040 Arc tangent to provious path: G08 (2nd part)
Page 1041 Arc defined by three points: G09 (1st part)
Page 1042 Arc defined by three points: G09 (2nd part)
Page 1043 Threadcutting: G33
Page 1044 Controlled corner rounding: G36
Page 1045 Tangential entry: G37
Page 1046 Tangential exit: G38
Page 1047 Chamfer blend: G39
Page 1048 Dwell/Block preparation stop: G04, G04K.
Page 1049 Round/Square corner: G05, G07.
Page 1050 Mirror image: G11, G12, G13, G14.
Page 1051 Planes and longitudinal axis selection: G15, G16, G17, G18, G19.
Page 1052 Work zones: G21, G22.
Page 1053 Tool radius compensation: G40,G41,G42.
Page 1054 Tool length compensation: G43,G44.
Page 1055 Zero offsets.
Page 1056 Millimeters/inches: G71, G70.
Page 1057 Scaling factor: G72.
Page 1058 Pattern rotation: G73.
Page 1059 Machine reference search: G74
Page 1060 Probing: G75.
Page 1061 Slaved axis: G77, G78.
Page 1062 Absolute/incremental programming: G90, G91.
Page 1063 Coordinate and polar origin preset: G92,G93.
Page 1064 Feedrate programming: G94,G95.
Page 1065 G functions associated with canned cycles: G79, G80, G98 and G99.
Page 1066 Auxiliary function programming F,S,T and D.
Page 1067 Auxiliary function M programming.

SYNTAX ASSISTANCE: CNC TABLES

Page 1090 Tool Offset table.
Page 1091 Tool table
Page 1092 Tool magazine table.
Page 1093 Miscellaneous (auxiliary) function M table.
Page 1094 Zero offset table.
Page 1095 Leadscrew error compensation tables.
Page 1096 Cross compensation table.
Page 1097 Machine parameter tables.
Page 1098 User parameter tables.
Page 1099 Password table.

SYNTAX ASSISTANCE: HIGH LEVEL

Page 1100 : ERROR and MSG mnemonics.
Page 1101 : GOTO and RPT mnemonics.
Page 1102 : OPEN and WRITE mnemonics.
Page 1103 : SUB and RET mnemonics.
Page 1104 : CALL, PCALL, MCALL, MDOFF and PROBE mnemonics.
Page 1105 : DSBLK, ESBLK, DSTOP, ESTOP, DFHOLD, EFHOLD mnemonics.
Page 1106 : IF statement.
Page 1107 : Assignment blocks.
Page 1108 : Mathematical expressions.
Page 1109 : PAGE mnemonic.
Page 1110 : ODW mnemonic.
Page 1111 : DW mnemonic.
Page 1112 : IB mnemonic.
Page 1113 : SK mnemonic.
Page 1114 : WKEY and SYSTEM mnemonics.
Page 1115 : KEYSRC mnemonic.
Page 1116 : WBUF mnemonic.
Page 1117 : SYMBOL mnemonic.

SYNTAX ASSISTANCE: CANNED CYCLES

Page 1070 : Straight line pattern canned cycle: G60.
Page 1071 : Rectangular pattern canned cycle: G61.
Page 1072 : Grid pattern canned cycle: G62.
Page 1073 : Circular pattern canned cycle: G63.
Page 1074 : Arc pattern canned cycle: G64.
Page 1075 : Arc-chord pattern canned cycle: G65.
Page 1076 : Irregular pocket (with islands) canned cycle: G66.
Page 1077 : Irregular pocket roughing cycle: G67.
Page 1078 : Irregular pocket finishing cycle: G68.
Page 1079 : Complex deep hole drilling cycle: G69.
Page 1080 : Drilling cycle: G81.
Page 1081 : Drilling cycle with dwell: G82.
Page 1082 : Simple deep hole drilling cycle: G83
Page 1083 : Tapping cycle: G84
Page 1084 : Reaming cycle: G85.
Page 1085 : Boring cycle with withdrawal in G00: G86.
Page 1086 : Rectangular pocket canned cycle: G87.
Page 1087 : Circular pocket canned cycle: G88.
Page 1088 : Boring cycle with withdrawal in G01: G89.

H MAINTENANCE

Cleaning:

The accumulated dirt inside the unit may act as a screen preventing the proper dissipation of the heat generated by the internal circuitry which could result in a harmful overheating of the CNC and, consequently, possible malfunctions.

On the other hand, accumulated dirt can sometimes act as an electrical conductor and shortcircuit the internal circuitry, especially under high humidity conditions.

To clean the operator panel and the monitor, a smooth cloth should be used which has been dipped into de-ionized water and /or non abrasive dish-washer soap (liquid, never powder) or 75º alcohol.

Do not use highly compressed air to clean the unit because it could generate electrostatic discharges.

The plastics used on the front panel are resistant to :

1.- Grease and mineral oils
2.- Bases and bleach
3.- Dissolved detergents
4.- Alcohol

Avoid the action of solvents such as Chlorine hydrocarbons , Benzole , Esters and Ether which can damage the plastics used to make the unit's front panel.

Preventive Inspection:

If the CNC does not turn on when actuating the start-up switch, verify that the monitor fuses are in good condition and that they are the right ones.

To check the fuses, first disconnect the power to the CNC.

Do not open this unit.
Only personnel authorized by Fagor Automation may open this module.

Do not handle the connectors with the unit connected to main AC power.
Before handling these connectors, make sure that the unit is not connected to main AC power.

Note :
Fagor Automation shall not be held responsible for any material or physical damage derived from the violation of these basic safety requirements.

List of materials, parts that could be replaced

		Code
Central Unit	3 modules	83060100
	6 modules	83060101
CPU module	Mill	83090122
	Lathe	83090123
	Sercos board	83160110
Axes module		83150100
I/O module		83210100
I/O Tracing module		83220100
Sercos module		83160100
Cover (empty module)		83300100
CPU Turbo		80500077

9" Amber monitor (no keyboard)	MC & TC	83390002
9" Amber monitor (with keyboard)	Mill	83390000
	Lathe	83390001
10" Color monitor (no keyboard)	MC & TC	83390004
10" Color monitor (with keyboard)	Mill	83420001
	Lathe	83420003
11" LCD monitor (no keyboard)	MC & TC	83480100
11" LCD Monitor (with keyboard)	Mill	83480101
	Lathe	83480102
	M & MC	83480103
	T & TC	83480104
14" Color monitor (no keyboard)		83390003
14" Color monitor (with keyboard)		83420004

		Código
Operator panel (no handwheel)	Mill	80300010
	Lathe	80300011
Operator panel (with handwheel)	Mill	80300014
	Lathe	80300015
Operator panel	MC	83540020
	TC	83540002
switcher board		83900000

Video cables	5m	83540020
	10m	83630021
	15m	83630022
	20m	83630023
	25m	83630024
Keyboard cables	2m	83630010
	5m	83630004
	10m	83630005
	15m	83630006
	20m	83630008
	25m	83630026

Configuration card MemKey Card	4 Mb	83120150
	8 Mb	83120160
	16 Mb	83120161
	24 Mb	83120162

Video adapter (digital - analog)		8C401001
Video duplicator	(DVD)	83900001
DNC software		80500115

Available manuals

Mill Model		Standard software (code)	Advanced software (code)
OEM Manuals	Spanish	03753400	03753460
	English	03753401	03753461
	French		
	German		
	Italian		
	portuguese		
User Manuals	Spanish	03753410	03753470
	English	03753411	03753471
	French	03753412	03753472
	German	03753413	03753473
	Italian	03753414	03753474
	portuguese	03753415	03753475
Conversational model (MC)	Spanish	03753440	03753500
	English	03753441	03753501
	French	03753442	03753502
	German	03753443	03753503
	Italian	03753444	03753504
	portuguese	03753445	03753505

Lathe Model		Standard software (code)	Advanced software (code)
OEM Manuals	Spanish	03753420	03753480
	English	03753421	03753481
	French		
	German		
	Italian		
	portuguese		
User Manuals	Spanish	03753430	03753490
	English	03753431	03753491
	French	03753432	03753492
	German	03753433	03753493
	Italian	03753434	03753494
	portuguese	03753435	03753495
Conversational model (TC)	Spanish	03753450	03753510
	English	03753451	03753511
	French	03753452	03753512
	German	03753453	03753511
	Italian	03753454	03753514
	portuguese	03753455	03753515

8055M CNC

ERROR TROUBLESHOOTING MANUAL

Ref. 9905 (ing)

INDEX

Programming errors **1**
(0001-0255)

Preparation and execution errors **34**
(1000-1238)

Hardware errors **52**
(2000-2028)

PLC errors **55**
(3000-3004)

Drive errors **56**
(4000-4025)

Table data errors **58**

Errors in 8055MC operating mode **61**

Alphabetical index **71**

PROGRAMMING ERRORS

0001 'Empty line.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE/S The various causes might be:

1.- When trying to enter into a program or execute an empty block or containing the label (block number).

2.- Within the «Irregular pocket canned cycle with islands (G66)», when parameter "S" (beginning of the profile) is greater than parameter "E" (end of profile).

SOLUTION The solution for each cause is:

1.- The CNC cannot enter into the program or execute an empty line. To do that, use the «;» symbol at the beginning of that block. The CNC will ignore the rest of the block.

2.- The value of parameter "S" (block where the profile definition begins) must be lower than the value of parameter "E" (block where the profile definition ends).

0002 'Improper data'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE/S The various causes might be:

1.- When editing an axis coordinate after the cutting conditions (F, S, T or D) or the «M» functions.

2.- When the marks of the block skip (conditional block /1, /2 or /3) are not at the beginning of the block.

3.- When programming a block number greater than 9999 while programming in ISO code.

4.- When trying to define the coordinates of the machining starting point in the finishing operation (G68) of the «Irregular pocket canned cycle».

5.- While programming in high-level, the value of the RPT instruction exceeds 9999.

SOLUTION The solution for each cause is:

1/2.- Remember that the programming order is:

1.- Block skip (conditional block /1, /2 or /3).
2.- Label (N).
3.- «G» functions.
4.- Axes coordinates (X, Y, Z…).
5.- Machining conditions (F, S, T, D).
6.- «M» functions.

All the data need not be programmed.

3.- Correct the block syntax. Program the labels between 0 and 9999

4.- No point can be programmed within the definition of the finishing cycle (G68) for the «Irregular pocket canned cycle». The CNC selects the point where it will start machining. The programming format is:

G68 B— L— Q— I— R— K— V—

And then the cutting conditions.

5.- Correct the block syntax. Program the labels between 0 and 9999

0003 'Improper data order.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE The machining conditions or the tool data have been programmed in the wrong order.

SOLUTION Remember that the programming order is:

… F— S— T— D— …

All the data need not be programmed.

0004 'No more information allowed in the block.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE/S The various causes might be:

1.- When editing a «G» function after an axis coordinate.

2.- When trying to edit some data after a «G» function (or after its associated parameters) which must go alone in the block (or which only admits its own associated data).

3.- When assigning a numeric value to a parameter that does not need it.

SOLUTION The solution for each cause is:

1.- Remember that the programming order is:

1.- Block skip (conditional block /1, /2 or /3).
2.- Label (N).
3.- «G» functions.
4.- Axes coordinates. (X, Y, Z…).
5.- Machining conditions (F, S, T, D).
6.- «M» functions.

All the data need not be programmed.

2.- There are some «G» functions which carry associated data in the block. Maybe, this type of functions do not let program other type of information after their associated parameters. On the other hand, neither machining conditions, (F, S), tool data (T, D) nor «M» functions may be programmed.

3.- There are some «G» functions having certain parameters associated to them which do not need to be defined with values.

0005 'Repeated information'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE The same data has been entered twice in a block.

SOLUTION Correct the syntax of the block. The same data cannot be defined twice in a block.

0006 'Improper data format'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE While defining the parameters of a machining canned cycle, a negative value has been assigned to a parameter which only admits positive values.

SOLUTION Verify the format of the canned cycle. In some canned cycles, there are parameters which only accept positive values.

0007 'Incompatible G functions.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE/S The various causes might be:

1.- When programming in the same block two «G» functions which are incompatible with each other.

2.- When trying to define a canned cycle in a block containing a nonlinear movement (G02, G03, G08, G09, G33).

SOLUTION The solution for each cause is:

1.- There are groups of «G» functions which cannot go together in the block because they involve actions incompatible with each other. For example:

G01/G02: Linear and circular interpolation
G41/G42: Left-hand or right-hand tool radius compensation.

This type of functions must be programmed in different blocks.

2.- A canned cycle must be defined in a block containing a linear movement. In other words, to define a cycle, a "G00" or a "G01" must be active. Nonlinear movements (G02, G03, G08 and G09) may be defined in the blocks following the profile definition.

0008 'Nonexistent G function'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE A nonexistent «G» function has been programmed.

SOLUTION Check the syntax of the block and verify that a different «G» function is not being edited by mistake.

0009 'No more G functions allowed in the block'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE A «G» function has been programmed after the machining conditions or after the tool data.

SOLUTION Remember that the programming order is:

1.- Block skip (conditional block /1, /2 or /3).
2.- Label (N).
3.- «G» functions.
4.- Axes coordinates. (X, Y, Z…).
5.- Machining conditions (F, S, T, D).
6.- «M» functions.

All the data need not be programmed.

0010 'No more M functions allowed in the block'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE More than 7 «M» functions have been programmed in a block.

SOLUTION The CNC does not let program more than 7 «M» functions in a block. To do so, write them in a separate block. The «M» functions may go alone in a block.

0011 'This G or M function must be alone.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE The block contains either a «G» or an «M» function that must go alone in the block.

SOLUTION Write it alone in the block.

0012 'Program F, S, T, D before the M functions.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE A machining condition (F, S) or tool data (T, D) has been programmed after the «M» functions.

SOLUTION Remember that the programming order is:

… F— S— T— D— M—

Up to 7 «M» functions may be programmed .
All the data need not be programmed.

0014 'Do not program labels by parameters.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE A label (block number) has been defined with a parameter.

SOLUTION The programming of a block number is optional, but it cannot be defined with a parameter, only with a number between 0 and 9999.

0015 'Number of repetitions not possible.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE A repetition has been programmed wrong or the block does not admit repetitions.

SOLUTION High level instructions do not admit a number of repetitions at the end of the block. To do a repetition, assign to the block to be repeated a label (block number) and use the RPT instruction.

0016 'Program: G15 axis.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE In the function «Longitudinal axis selection (G15)» the parameter for the axis has not been programmed.

SOLUTION Check the syntax of the block. The definition of the "G15" function requires the name of the new longitudinal axis.

0017 'Program: G16 axis-axis.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE In the function «Main plane selection by two axes (G16)» one of the two parameters for the axes has not been programmed.

SOLUTION Check the syntax of the block. The definition of the "G16" function requires the name of the axes defining the new work plane.

0018 'Program: G22 K(1/2/3/4) S(0/1/2).'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE In the function «Enable/Disable work zones (G22)» the type of enable or disable of the work zone has not been defined or it has been assigned the wrong value.

SOLUTION The parameter for enabling or disabling the work zones "S" must always be programmed and it may take the following values.

- S=0: The work zone is disabled.
- S=1: It is enabled as a no-entry zone.
- S=2: It is enabled as a no-exit zone.

0019 'Program: work zone K1, K2, K3 or K4.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE/S The various causes might be:

1.- A "G20", "G21" or "G22" function has been programmed without defining the work zone K1, K2, K3 or K4.

2.- The programmed work zone is smaller than 0 or greater than 4.

SOLUTION The solution for each cause is:

1.- The programming format for functions "G20", "G21" and "G22" is:

G20 K— X...C±5.5 (Definition of lower work zone limits).
G21 K— X...C±5.5 (Definition of upper work zone limits).
G22 K— S— (Enable/disable work zones).

Where:

- K : Is the work zone.
- X...C : Are the axes where the limits are defined.
- S : Is the type of work zone enable.

2.- The "K" work zone may only have the values of K1, K2, K3 or K4.

0020 'Program G36-G39 with R+5.5.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE In the "G36" or "G39" function, the "R" parameter has not been programmed or it has been assigned a negative value.

SOLUTION To define "G36" or "G39", parameter "R" must also be defined and with a positive value).

G36: R= Rounding radius.
G39: R= Distance between the end of the programmed path and the point to be chamfered.

0021 'Program: G72 S5.5 or axes.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE/S The various causes might be:
1.- When programming a general scaling factor (G72) without the scaling factor to apply.
2.- When programming a particular scaling factor (G72) to several axes, but the axes have been defined in the wrong order.

SOLUTION Remember that this function must be programmed in the following order:
- "G72 S5.5" When applying a general scaling factor (to all axes).
- "G72 X…C5.5" When applying a particular scaling factor to one or several axes.

0022 'Program: G73 Q (angle) I J (center).'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE/S The parameters of the «Pattern rotation (G73)» function have been programmed wrong. The causes may be:
1.- The rotation angle has not been defined.
2.- Only one of the rotation center coordinates has been defined.
3.- The rotation center coordinates have been defined in the wrong order.

SOLUTION The programming format for this function is:
G73 Q (angle) [I J] (center)
The "Q" value must always be programmed.
The "I", "J" values are optional, but if programmed, both must be programmed.

0023 'Block incompatible when defining a profile.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE In the set of blocks defining a pocket profile, there is a block containing a «G» function that cannot be part of the profile definition.

SOLUTION The "G" functions available in the profile definition of a pocket (2D/3D) are:
G00: Beginning of the profile.
G01: Linear interpolation.
G02/G03: Clockwise/counterclockwise interpolation.
G06: Circle center in absolute coordinates.
G08: Arc tangent to previous path.
G09: Three point arc.
G36: Controlled corner rounding
G39: Chamfer.
G53: Programming with respect to home.
G70/G71: Inch/metric programming.
G90/G91: Programming in absolute/incremental coordinates.
G93: Polar origin preset.
And also, in the 3D pocket profile:
G16: Main plane selection by two axes.
G17: Main plane X-Y and longitudinal Z.
G18: Main plane Z-X and longitudinal Y.
G19: Main plane Y-Z and longitudinal X.

0024 'High level blocks not allowed when defining a profile.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE Within the set of blocks defining a pocket profile, a high level block has been programmed.

SOLUTION The pocket profile must be defined in ISO code. High level instructions are not allowed (GOTO, MSG, RPT ...).

0025 'Program: G77 axes (2 thru 6).'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE In the «Axis slaving (G77)» function, the parameters for the axes have not been programmed.

SOLUTION The programming of "G77" function requires at least two axes.

0026 'Program: G93 I J.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE In the «Polar origin preset (G93)» function, some of the parameters for the new polar origin have not been programmed.

SOLUTION Remember that the programming format for this function is:

G93 I— J—

The "I", "J" values are optional, but if programmed, both must be programmed and they indicate the new polar origin.

0027 'G49 T X Y Z S, X Y Z A B C ', or, ' X Y Z Q R S.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE In the «Incline plane definition (G49)» function, a parameter has been programmed twice.

SOLUTION Check the syntax of the block. The programming formats are:

T X Y Z S X Y Z A B C X Y Z Q R S

0028 'G2 or G3 not allowed when programming a canned cycle.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE A canned cycle has been attempted to execute while the "G02", "G03" or "G33" functions were active.

SOLUTION To execute a canned cycle, "G00" or "G01" must be active. Maybe, a "G02" or "G03" function was activated in the M code history instead. Check that these functions are not active when the canned cycle is defined.

0029 'G60: [A] /X I K/(2) [P Q R S T U V].'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE The parameters of the «Multiple machining in a straight line (G60)» have been programmed wrong. These are the possible causes:

1.- Some mandatory parameter is missing.
2.- The parameters of the cycle have not been edited in the correct order.
3.- Some data might be superfluous.

SOLUTION In this type of machining, two of the following parameters must always be programmed:

X : Path length.
I : Step between machining operations.
K : Number of machining operations.

The rest of the parameters are optional. The parameters must be programmed in the order shown by the error message.

0030 'G61-2: [A B] /X I J/(2) Y J D (2)/ [P Q R S T U V].'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE The parameters of the «Multiple machining in a parallelogram pattern (G61)» or «Multiple machining in a grid pattern (G62)» cycle have been programmed wrong. These may be the probable causes:
1.- Some mandatory parameter is missing.
2.- The parameters of the cycle have not been edited in the correct order.
3.- Some data might be superfluous.

SOLUTION This type of machining requires the programming of two parameters of each group (X, I, K) and (Y, J, D).
X/Y : Path length.
I/J : Step between machining operations.
K /D : Number of machining operations.
The rest of the parameters are optional. The parameters must be edited in the order indicated by the error message.

0031 'G63: X Y /I K/(1) [C P][P Q R S T U V].'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE The parameters of the «Multiple machining in a circle (G63)» cycle have been programmed wrong. These may be the probable causes:
1.- Some mandatory parameter is missing.
2.- The parameters of the cycle have not been edited in the correct order.
3.- Some data might be superfluous.

SOLUTION This type of machining requires the programming of:
X/Y : Distance from the center to the first hole.
And one of the following data:
I : Angular step between machining operations.
K : Number of machining operations.
The rest of the parameters are optional. The parameters must be edited in the order indicated by the error message.

0032 'G64: X Y /I K/(1) [C P][P Q R S T U V.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE The parameters of the «multiple machining in an arc (G64)» cycle have been programmed wrong. These may be the probable causes:
1.- Some mandatory parameter is missing.
2.- The parameters of the cycle have not been edited in the correct order.
3.- Some data might be superfluous.

SOLUTION This type of machining requires the programming of:
X/Y : Distance from the center to the first hole.
B : Total angular travel.
And one of the following data:
I : Angular step between machining operations.
K : Number of machining operations.
The rest of the parameters are optional. The parameters must be edited in the order indicated by the error message.

0033 'G65: X Y /A I/(1) [C P].'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE The parameters of the «Multiple machining programmed by means of an arc chord (G65)» cycle have been programmed wrong. These may be the probable causes:
1.- Some mandatory parameter is missing.
2.- The parameters of the cycle have not been edited in the correct order.
3.- Some data might be superfluous.

SOLUTION This type of machining requires the programming of:
X/Y : Distance from the center to the first hole.
And one of the following data:
A : Angle of the matrix of the chord with the abscissa axis (in degrees).
I : Chord length.
The rest of the parameters are optional. The parameters must be edited in the order indicated by the error message.

0034 'G66: [D H][R I][C J][F K] S E [Q].'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE The parameters of the «Irregular pocket canned cycle with islands (G66)» have been programmed wrong. These may be the probable causes:
1.- Some mandatory parameter is missing.
2.- The parameters of the cycle have not been edited in the correct order.
3.- Some data might be superfluous.

SOLUTION This machining cycle requires the programming of :
S : First block of the description of the geometry of the profiles making up the pocket.
E : End block of the description of the geometry of the profiles making up the pocket.
The rest of the parameters are optional. The parameters must be edited in the order indicated by the error message. Also, the following parameters cannot be defined:
H if D has not been defined.
I if R has not been defined.
J if C has not been defined.
K if F has not been defined.
The (X...C) position where the machining takes place cannot be programmed either.

0035 'G67: [A] B [C] [I] [R] [K] [V].'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE The parameters of the roughing (2D/3D pocket) or semi-finishing (3D pocket) operation have been programmed wrong in the «Irregular pocket canned cycle with islands». These may be the probable causes:
1.- Some mandatory parameter is missing.
2.- The parameters of the cycle have not been edited in the correct order.
3.- Some data might be superfluous.

SOLUTION This machining cycle requires the programming of :
ROUGING OPERATION (2D or 3D pockets)
B : Machining pass.
I : Total pocket depth.
R : Coordinate of the reference plane.
SEMI-FINISHING OPERATION (3D pockets)
B : Machining pass.
I : Total pocket depth (if no roughing operation has been defined).
R : Coordinate of the reference plane (if no roughing operation has been defined).
The rest of the parameters are optional. The parameters must be edited in the order indicated by the error message. The (X...C) position where the machining takes place cannot be programmed in this cycle.

0036 'G68: [B] [L] [Q] [J] [I] [R] [K].'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE The parameters for the finishing operation (2D/3D pocket) have been programmed wrong in the «Irregular pocket cycle with islands. These may be the probable causes:
1.- A parameter has been programmed which does not match the calling format.
2.- Some mandatory parameter is missing.
3.- The parameters of the cycle have not been edited in the correct order.

SOLUTION This machining cycle requires the programming of :

2D pockets

B : Cutting pass (if no roughing operation has been defined).
I : Total pocket depth (if no roughing operation has been defined).
R : Coordinate of the reference plane (if no roughing operation has been defined).

3D pockets

B : Cutting pass
I : Total pocket depth (if no roughing or semi-finishing operation has been defined).
R : Coordinate of the reference plane (if no roughing or semi-finishing operation has been defined).

The rest of the parameters are optional. The parameters must be edited in the order indicated by the error message. The (X...C) position where the machining takes place cannot be programmed in this cycle.

0037 'G69: I B [C D H J K L R].'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE The parameters of the «Deep hole drilling cycle with variable peck (G69)». These may be the probable causes:
1.- Some mandatory parameter is missing.
2.- The parameters of the cycle have not been edited in the correct order.

SOLUTION This type of machining requires the programming of:
I : Machining depth.
B : Drilling peck.
The rest of the parameters are optional. The parameters must be edited in the order indicated by the error message. The (X...C) position where the machining takes place can be programmed in this cycle.

0038 'G81-84-85-86-89: I [K].'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE The parameters have been programmed wrong in the following cycles: drilling (G81), tapping (G84), reaming (G85) or boring (G86/G89). This could be because parameter "I : Machining depth" is missing in the canned cycle being edited.

SOLUTION This type of machining requires the programming of:
I : Machining depth.
The rest of the parameters are optional. The parameters must be edited in the order indicated by the error message. The (X...C) position where the machining takes place can be programmed in this cycle.

0039 'G82: I K.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE The parameters have been programmed wrong in the «Drilling cycle with dwell (G82)». This could be because some parameter is missing.

SOLUTION Both parameters must be programmed in this cycle:
I : Machining depth.
K : Dwell at the bottom.
To program a drilling operation without dwell at the bottom, use function G81.
The parameters must be edited in the order indicated by the error message. The (X...C) position where the machining takes place can be programmed in this cycle.

0044 'Incorrect Coordinates.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE The "I : Machining depth" parameter is missing in the definition of a machining canned cycle (G81-G89)

SOLUTION This type of machining requires the programming of:

I : Machining depth.

The rest of the parameters are optional. The parameters must be edited in the order indicated by the error message. The (X...C) position where the machining takes place can be programmed in this cycle.

0045 'Polar coordinates not allowed.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE When «Programming with respect to home (G53)», the end point has been defined in polar or cylindrical coordinates or in Cartesian coordinates with an angle.

SOLUTION When programming with respect to home, only Cartesian coordinates may be programmed.

0046 'Axis does not exist.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE/S The various causes might be:

1.- When editing a block whose execution involves the movement of a nonexistent axis.

2.- Sometimes, this error comes up while editing a block that is missing a parameter of the «G» function. This is because some parameters with an axis name have a special meaning inside certain «G» functions. For example:

G69 I— B—.

In this case, parameter "B" has a special meaning after "I". If the "I" parameter is left out, the CNC assumes "B" as the position where the machining takes place on that axis. If that axis does not exist, it will issue this error message.

SOLUTION The solution for each cause is:

1.- Check that the axis name being edited is correct.

2.- Check the block syntax and make sure that all the mandatory parameters have been programmed.

0047 'Program axes.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE No axis has been programmed in a function requiring an axis.

SOLUTION Some instructions require the programming of axes (REPOS, G14, G20, G21…).

0048 'Incorrect order of axes.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE The axis coordinates have not been programmed in the correct order or an axis has been programmed twice in the same block.

SOLUTION Remember that the correct programming order for the axes is:

X— Y— Z— U— V— W— A— B— C—

All axes need not be programmed:

0049 'Point incompatible with active plane.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE/S The various causes might be:
1.- When trying to do a circular interpolation, the end point is not in the active plane.
2.- When trying to do a tangential exit in a path that is not in the active plane.

SOLUTION The solution for each cause is:
1.- Maybe a plane has been defined with "G16", "G17", "G18" or "G19". In this case, circular interpolations can only be carried out on the main axes defining that plane. To define a circular interpolation in another plane, it must be defined beforehand.
2.- Maybe a plane has been defined with "G16", "G17", "G18" or "G19". In this case, corner rounding, chamfers and tangential entries/exits can only be carried out on the main axes defining that plane. To do it in another plane, it must be defined beforehand.

0053 'Program pitch.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE In the «Electronic threading cycle (G33)» the parameter for the thread pitch is missing.

SOLUTION Remember that the programming format for this function is:

G33 X...C— L—

Where:
L : Is the thread pitch.

0054 'Pitch programmed incorrectly.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE A helical interpolation has been programmed with the wrong or negative pitch.

SOLUTION Remember that the programming format is:

G02/G03 X— Y— I— J— Z— K—

Where:
K : is the helical pitch (always positive value).

0057 'Do not program a slaved axis.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE The various causes might be:
1.- When trying to move an axis alone while being slaved to another one.
2.- When trying to slave an axis that is already slaved using the G77 function «Electronic axis slaving».

SOLUTION The solution for each cause is:
1.- A slaved axis cannot be moved separately. To move a slaved axis, its master axis must be moved. Both axes will move at the same time.

Example: If the Y axis is slaved to the X axis, an X axis move must be programmed in order to move the Y axis (together with the X axis).

To unslave the axis, program "G78".
2.- An axis cannot be slaved to two different axes at the same time. To unslave the axes, program "G78".

0058 'Do not program a GANTRY axis.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE/S The various causes might be:
1.- When trying to move an axis alone while being slaved to another one as a GANTRY axis
2.- When defining an operation on a GANTRY axis. (Definition of work zone limits, planes, etc.).

SOLUTION The solution for each cause is:
1.- A GANTRY axis cannot be moved separately. To move a GANTRY axis, its associated axis must be moved. Both axes will move at the same time.
Example: If the Y axis is a GANTRY axis associated with the X axis, an X axis move must be programmed in order to move the Y axis (together with the X axis).
GANTRY axes are defined by machine parameter.
2.- The axes defined as GANTRY cannot be used in the definition of operations or movements. These operations are defined with the main axis that the GANTRY axis is associated with.

0059 'HIRTH axis: program only integer values.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE A rotation of a HIRTH axis has been programmed with a decimal value.

SOLUTION HIRTH axes do not accept decimal angular values. They must be full degrees.

0061 'ELSE not associated with IF.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE/S The various causes might be:
1.- While editing in High level language, when editing the "ELSE" instruction without having previously programmed an "IF".
2.- When programming in high level language, an "IF" is programmed without associating it with any action after the condition.

SOLUTION Remember that the programming formats for this instruction are:
(IF *(condition)* <*action1*>*)*
(IF *(condition)* <*action1*> ELSE <*action2*>)
If the *condition* is true, it executes the <*action1*>, otherwise, it executes the <*action2*>.

0062 'Program label N(0-9999).'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE While programming in high level language, a block number out of the 0-9999 range has been programmed in the "RPT" or "GOTO" instruction.

SOLUTION Remember that the programming formats for these instructions are:
(RPT N(block number), N(block number))
(GOTO N(block number))
The block number (label) must be between 0 and 9999.

0063 'Program subroutine number 1 thru 9999.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE While programming in high level language, a subroutine number out of the 0-9999 range has been programmed in the "SUB" instruction.

SOLUTION Remember that the programming format for this instruction is:
(SUB (integer))
The subroutine number must be between 0 and 9999.

0064 'Repeated subroutine.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE There has been an attempt to define a subroutine already existing in another program of the memory.

SOLUTION In the CNC memory, there could not be more than one subroutine with the same identifying number even if they are contained in different programs.

0065 'The main program cannot have a subroutine.'

DETECTED In execution or while executing programs transmitted via DNC.

CAUSE/S The various causes might be:
1.- An attempt has been made to define a subroutine in the MDI execution mode.
2.- A subroutine has been defined in the main program.

SOLUTION The solution for each cause is:
1.- Subroutines cannot be defined from the «MDI execution» option of the menu.
2.- Subroutines must be defined after the main program or in a separate program. They cannot be defined before or inside the main program.

0066 'Expecting a message.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE While programming in high level, the "MSG" or "ERROR" instruction has been edited but without the message to be displayed.

SOLUTION Remember that the programming format of these instructions is:
(MSG "message")
(ERROR integer, "error message")
Although it can also be programmed like:
(ERROR integer)
(ERROR "error message")

0067 'OPEN is missing.'

DETECTED In execution or while executing programs transmitted via DNC.

CAUSE While programming in high level, a "WRITE" instruction has been edited, but the OPEN instruction has not been written previously to tell it where that instruction has to be executed.

SOLUTION The "OPEN" instruction must be edited before the "WRITE" instruction to «tell» the CNC where (in which program) it must execute the "WRITE" instruction.

0069 'Program does not exist.'

DETECTED In execution or while executing programs transmitted via DNC.

CAUSE Inside the «Irregular pocket with islands cycle (G66)», it has been programmed that the profiles defining the irregular pocket are in another program (parameter "Q"), but that program does not exist.

SOLUTION Parameter "Q" defines which program contains the definition of the profiles that, in turn, define the irregular pocket with islands. If this parameter is programmed, that program number must exist and it must contain the labels defined by parameters "S" and "E".

0070 'Program already exists.'

DETECTED In execution or while executing programs transmitted via DNC.

CAUSE This error comes up during execution when using the "OPEN" instruction (While programming in high level language) to create an already existing program.

SOLUTION Change the program number or use parameters A/D in the "OPEN" instruction:
(OPEN P———,A/D,…)
Where:
- A: Appends new blocks after the existing ones.
- D: Deletes the existing program and it opens it as a new one.

0071 'Expecting a parameter'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE/S The various causes might be:

1.- When defining the function «Modification of canned cycle parameters (G79)», the parameter to be modified has not been indicated.

2.- While editing the machine parameter table, the wrong parameter number has been entered (maybe the "P" character is missing) or another action is being carried out (moving around in the table) before quitting the table editing mode.

SOLUTION The solution for each cause is:

1.- To define the "G79" function, the cycle parameter to be modified must be indicated as well as its new value.

2.- Enter the parameter number to be edited or press [ESC] to quit this mode.

0072 'Parameter does not exist.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE While programming in high level language, the "ERROR" instruction has been edited, but the error number to be displayed has been defined either with a local parameter greater than 25 or with a global parameter greater than 299.

SOLUTION The parameters used by the CNC are:

- Local: 0-25
-Global: 100-299

0075 'Read-only variable.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE An attempt has been made to assign a value to a read-only variable.

SOLUTION Read-only variables cannot be assigned any values through programming. However, their values can be assigned to a parameter.

0077 'Analog output not available.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE An attempt has been made to write to an analog output currently being used by the CNC.

SOLUTION The selected analog output may be currently used by an axis or a spindle. Select another analog output between 1 and 8.

0078 'Program channel 0(CNC),1(PLC) or 2(DNC).'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE While programming in high level language, the "KEYSCR" instruction has been programmed, but the source of the keys is missing.

SOLUTION When programming the "KEYSCR" instruction, the parameter for the source of the keys must always be programmed:

(KEYSCR=0) : CNC keyboard
(KEYSCR=1) : PLC
(KEYSCR=2) : DNC

The CNC only lets modifying the contents of this variable if it is «zero»

0079 'Program error number 0 thru 9999.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE While programming in high level language, the "ERROR" instruction has been programmed, but the error number to be displayed is missing.

SOLUTION Remember that the programming format for this instruction is:
(ERROR integer, "error message")
Although it can also be programmed as follows:
(ERROR integer)
(ERROR "error message")

0081 'Incorrect expression.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE While programming in high level language, an expression has been edited with the wrong format.

SOLUTION Correct the block syntax.

0082 'Incorrect operation.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE/S The various causes might be:
1.- While programming in high level language, the assignment of a value to a parameter is incomplete.
2.- While programming in high level language, the call to a subroutine is incomplete.

SOLUTION Correct (complete) the format to assign a value to a parameter or a call to a subroutine.

0083 'Incomplete operation.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE/S The various causes might be:
1.- While programming in high level language, the "IF" instruction has been edited without the condition between brackets.
2.- While programming in high level language, the "DIGIT" instruction has been edited without assigning a value to some parameter.

SOLUTION The solution for each cause is:
1.- Remember that the programming format for this instruction are:
(IF (*condition*) <*action1*>)
(IF (*condition*) <*action1*> ELSE <*action2*>)
If the *condition* is true, it executes the <*action1*>, otherwise, it executes <*action2*>.
2.- Correct the syntax of the block. All the parameters defined within the "DIGIT" instruction must have a value assigned to them.

0084 'Expecting "=".'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE While programming in high level language, a symbol or data has been entered that does not match the syntax of the block.

SOLUTION Enter the "=" symbol in the right place.

0085 'Expecting ")".'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE While programming in high level language, a symbol or data has been entered that does not match the syntax of the block.

SOLUTION Enter the ")" symbol in the right place.

0086 'Expecting "(".'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE While programming in high level language, a symbol or data has been entered that does not match the syntax of the block.

SOLUTION Enter the "(" symbol in the right place.

0087 'Expecting ",".'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE/S The various causes might be:

1.- While programming in high level language, a symbol or data has been entered that does not match the syntax of the block.
2.- While programming in high level language, an ISO-coded instruction has been programmed.
3.- While programming in high level language, an operation has been assigned either to a local parameter greater than 25 or to a global parameter greater 299.

SOLUTION The solution for each cause is:

1.- Enter the "," symbol in the right place.
2.- A block cannot contain high level language instructions and ISO-coded instructions at the same time.
3.- The parameters used by the CNC are:
- Local: 0-25.
- Global: 100-299.

Other parameters out of this range cannot be used in operations.

0089 'Logarithm of zero or negative number.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE An operation has been programmed which involves the calculation of a negative number or a zero.

SOLUTION Only logarithms of numbers greater than zero can be calculated. When working with parameters, that parameter may have already acquired a negative value or zero. Check that the parameter does not reach the operation with that value.

0090 'Square root of a negative number.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE An operation has been programmed which involves the calculation of the square root of a negative number.

SOLUTION Only the square root of numbers greater than zero can be calculated. When working with parameters, that parameter may have already acquired a negative value or zero. Check that the parameter does not reach the operation with that value.

0091 'Division by zero.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE An operation has been programmed which involves a division by zero.

SOLUTION Only divisions by numbers other than zero are allowed. When working with parameters, that parameter may have already acquired a negative value or zero. Check that the parameter does not reach the operation with that value.

0092 'Base zero with positive exponent.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE An operation has been programmed which involves elevating zero to a negative exponent (or zero).

SOLUTION Zero can only be elevated to positive exponents greater than zero. When working with parameters, that parameter may have already acquired a negative value or zero. Check that the parameter does not reach the operation with that value.

0093 'Negative base with decimal exponent.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE An operation has been programmed which involves elevating a negative number to a decimal exponent.

SOLUTION Negative numbers can only be elevated to integer exponents. When working with parameters, that parameter may have already acquired a negative value or zero. Check that the parameter does not reach the operation with that value.

0094 'ASIN/ACOS range exceeded.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE An operation has been programmed which involves calculating the arcsine or arccosine of a number out of the ±1 range.

SOLUTION Only the arcsine (ASIN) or arccosine (ACOS) of numbers between ±1 can be calculated. When working with parameters, that parameter may have already acquired a negative value or zero. Check that the parameter does not reach the operation with that value.

0095 'Program row number.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE While editing a customizing program, a window has been programmed with the "ODW" instruction, but the vertical position of the window on the screen is missing.

SOLUTION The vertical position of the window on the screen is defined by rows (0-25).

0096 'Program column number.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE While editing a customizing program, a window has been programmed with the "ODW" instruction, but the horizontal position of the window on the screen is missing.

SOLUTION The horizontal position of the window on the screen is defined by columns (0-79).

0097 'Program another softkey.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE While editing a customizing program, the programming format for the "SK" instruction has not been respected.

SOLUTION Correct the syntax of the block. The programming format is:

(SK1=(text 1), SK2=(text 2)…)

If the "," character is entered after a text, the CNC expects the name of another softkey.

0098 'Program softkeys 1 thru 7.'

DETECTED While executing in the user channel.

CAUSE In the block syntax, a softkey has been programmed out of the 1 to 7 range.

SOLUTION Only softkeys within the 1 to 7 range can be programmed.

0099 'Program another window.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE While editing a customizing program, the programming format for the "DW" instruction has not been respected.

SOLUTION Correct the syntax of the block. The programming format is:

(DW1=(assignment), DW2=(assignment)…)

If the "," character is entered after an assignment, the CNC expects the name of another window.

0100 'Program windows 0 thru 25.'

DETECTED While executing in the user channel.
CAUSE In the block syntax, a window has been programmed out of the 0 to 25 range.
SOLUTION Only windows within the 0 to 25 range can be programmed.

0101 'Program rows 0 thru 20.'

DETECTED While executing in the user channel.
CAUSE In the block syntax, a row has been programmed out of the 0 to 20 range.
SOLUTION Only rows within the 0 to 20 range can be programmed.

0102 'Program columns 0 thru 79.'

DETECTED While executing in the user channel.
CAUSE In the block syntax, a column has been programmed out of the 0 to 79 range.
SOLUTION Only columns within the 0 to 79 range can be programmed.

0103 'Program pages 0 thru 255.'

DETECTED While executing in the user channel.
CAUSE In the block syntax, a page has been programmed out of the 0 to 255 range.
SOLUTION Only pages within the 0 to 255 range can be programmed.

0104 'Program INPUT.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.
CAUSE While programming in high level language, an "IB" instruction has been edited without associating an "INPUT" to it.
SOLUTION Remember that the programming formats for this instruction are:
(IB (expression) = INPUT "text", format)
(IB (expression) = INPUT "text")

0105 'Program inputs 0 thru 25.'

DETECTED While executing in the user channel.
CAUSE In the block syntax, an input has been programmed out of the 0 to 25 range.
SOLUTION Only inputs within the 0 to 25 range can be programmed.

0106 'Program numerical format.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.
CAUSE While programming in high level language, an "IB" instruction has been edited with non-numeric format.
SOLUTION Remember that the programming format for this instruction is:
(IB (expression) = INPUT "text", format)
Where «format» must be a signed number with 6 entire digits and 5 decimals at the most.
If the "," character is entered after the text, the CNC expects the format.

0107 'Do not program formats greater than 6.5 .'

DETECTED While executing in the user channel.
CAUSE While programming in high level language, an "IB" instruction has been edited in a format with more than 6 entire digits or more than 5 decimals.
SOLUTION Remember that the programming format for this instruction is:
(IB (expression) = INPUT "text", format)
Where «format» must be a signed number with 6 entire digits and 5 decimals at the most.

0108 'This command can only be executed in the user channel.'

DETECTED During execution.

CAUSE An attempt has been made to execute a block containing information that can only be executed through the user channel.

SOLUTION There are specific expressions for customizing programs that can only be executed inside the user program.

0109 'User channel: Do not program geometric aides, comp. or cycles'

DETECTED While executing in the user channel.

CAUSE An attempt has been made to execute a block containing geometric aide, tool radius/length compensation or machining canned cycles.

SOLUTION Inside a customizing program the following cannot be programmed:
- Neither geometric assistance nor movements.
- Neither tool radius nor length compensation.
- Canned cycles.

0110 'Local parameters not allowed.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE Some functions can only be programmed with global parameters.

SOLUTION Global parameters are the ones included in the 100-299 range.

0111 'Block cannot be executed while running another program'

DETECTED While executing in MDI mode.

CAUSE An attempt has been made to execute a customizing instruction from MDI mode while the user channel program is running.

SOLUTION Customizing instructions can only be executed through the user channel.

0112 'WBUF can only be executed in user channel while editing'

DETECTED During execution or user channel execution.

CAUSE An attempt has been made to execute the "WBUF" instruction.

SOLUTION The "WBUF" instruction cannot be executed. It can only be used in the editing stage through the user input.

0113 'Table limits exceeded.'

DETECTED While editing tables.

CAUSE/S The various causes might be:
1.- In the tool offset table, an attempt has been made to define a tool offset with a greater number than allowed by the manufacturer.
2.- In the parameter tables, an attempt has been made to define a nonexistent parameter.

SOLUTION The tool offset number must be smaller than the one allowed by the manufacturer.

0114 'Offset: D3 R L I K.'

DETECTED While editing tables.

CAUSE In the tool offset table, the parameter editing order has not been respected.

SOLUTION Enter the table parameters in the right order.

0115 'Tool: T4 D3 F3 N5 R5(.2).'

DETECTED While editing tables.

CAUSE In the tool table, the parameter editing order has not been respected.

SOLUTION Enter the table parameters in the right order.

0116 'Zero offset: G54-59 axes (1-5).'

DETECTED While editing tables.

CAUSE In the Zero offset table, the zero offset to be defined (G54-G59) has not be selected.

SOLUTION Enter the table parameters in the right order. To fill out the zero offset table, first select the offset to be defined (G54-G59) and then the zero offset position for each axis.

0117 'M function: M4 S4 bits(8).'

DETECTED While editing tables.

CAUSE In the «M» function table, the parameter editing order has not been respected.

SOLUTION Edit table following the format:

M1234 (associated subroutine) (customizing bits)

0118 'G51 [A] E'

DETECTED In execution or while executing programs transmitted via DNC.

CAUSE In the «Look-Ahead (G51)» function, the parameter for the maximum contouring error is missing.

SOLUTION This type of machining requires the programming of:

E : Maximum contouring error.

The rest of the parameters are optional. The parameters must be edited in the order indicated by the error message.

0119 'Leadscrew: Position-Error.'

DETECTED While editing tables.

CAUSE In the leadscrew compensation tables, the parameter editing order has not been respected.

SOLUTION Enter the table parameters in the right order

P123 (position of the axis to be compensated) (leadscrew error at that point)

0120 'Incorrect axis.'

DETECTED While editing tables.

CAUSE In the leadscrew compensation tables, an attempt has been made to edit a different axis from the one corresponding to that table.

SOLUTION Each axis has its own table for leadscrew compensation. The table for each axis can only contain the positions for that axis.

0121 'Program P3 = value.'

DETECTED While editing tables.

CAUSE In the machine parameter table, the editing format has not been respected.

SOLUTION Enter the table parameters in the right order.

P123 = (parameter value)

0122 'Magazine: P(1-255) = T(1-9999).'

DETECTED While editing tables.

CAUSE In the tool magazine table, the editing format has not been respected or some data is missing.

SOLUTION Enter the table parameters in the right order.

0123 'Tool T0 does not exist.'

DETECTED While editing tables.

CAUSE In the tool table, an attempt has been made to edit a tool as T0.

SOLUTION No tool can be edited as T0. The first tool must be T1.

0124 'Offset D0 does not exist.'

DETECTED While editing tables.

CAUSE In the tool table, an attempt has been made to edit a tool offset as D0.

SOLUTION No tool offset can be edited as D0. The first tool offset must be D1.

0125 'Do not modify the active tool or the next one.'

DETECTED During execution.

CAUSE In the tool magazine table, an attempt has been made to change the active tool or the next one.

SOLUTION During execution, neither the active tool nor the next one may be changed.

0126 'Tool not defined.'

DETECTED While editing tables.

CAUSE In the tool magazine table, an attempt has been made to assign to the magazine position a tool that is not defined in the tool table.

SOLUTION Define the tool in the tool table.

0127 'Magazine is not RANDOM.'

DETECTED While editing tables.

CAUSE There is no RANDOM magazine and, in the tool magazine table, the tool number does not match the tool magazine position.

SOLUTION When the tool magazine is not RANDOM, the tool number must be the same as the magazine position (pocket number).

0128 'The position of a special tool is set.'

DETECTED While editing tables.

CAUSE In the tool magazine table, an attempt has been made to place a tool in a magazine position reserved for a special tool.

SOLUTION When a special tool occupies more than one position in the magazine, it has a reserved position in the magazine. No other tool can be placed in this position.

0129 'Next tool only possible in machining centers.'

DETECTED During execution.

CAUSE A tool change has been programmed with M06 and the machine is not a machining center (it is not expecting the next tool).

SOLUTION When the machining is not a machining center, the tool change is done automatically when programming the tool number «T».

0130 'Write 0/1.'

DETECTED While editing machine parameters

CAUSE An attempt has been made to assign the wrong value to a parameter.

SOLUTION The parameter only admits values of 0 or 1.

0131 'Write +/-.'

DETECTED While editing machine parameters
CAUSE An attempt has been made to assign the wrong value to a parameter.
SOLUTION The parameter only admits values of + or -.

0132 'Write YES/NO.'

DETECTED While editing machine parameters
CAUSE An attempt has been made to assign the wrong value to a parameter.
SOLUTION The parameter only admits values of YES or NO.

0133 'Write ON/OFF.'

DETECTED While editing machine parameters
CAUSE An attempt has been made to assign the wrong value to a parameter.
SOLUTION The parameter only admits values of ON or OFF.

0134 'Values 0 thru 2.'

DETECTED While editing machine parameters
CAUSE An attempt has been made to assign the wrong value to a parameter.
SOLUTION The parameter only admits values between 0 and 2.

0135 'Values 0 thru 3.'

DETECTED While editing machine parameters
CAUSE An attempt has been made to assign the wrong value to a parameter.
SOLUTION The parameter only admits values between 0 and 3.

0136 'Values 0 thru 4.'

DETECTED While editing machine parameters
CAUSE An attempt has been made to assign the wrong value to a parameter.
SOLUTION The parameter only admits values between 0 and 4.

0137 'Values 0 thru 9.'

DETECTED While editing machine parameters
CAUSE An attempt has been made to assign the wrong value to a parameter.
SOLUTION The parameter only admits values between 0 and 9.

0139 'Values 0 thru 100.'

DETECTED While editing machine parameters
CAUSE An attempt has been made to assign the wrong value to a parameter.
SOLUTION The parameter only admits values between 0 and 100.

0140 'Values 0 thru 255.'

DETECTED While editing machine parameters
CAUSE An attempt has been made to assign the wrong value to a parameter.
SOLUTION The parameter only admits values between 0 and 255.

0141 'Values 0 thru 9999.'

DETECTED While editing machine parameters

CAUSE/S The various causes might be:
1.- An attempt has been made to assign the wrong value to a parameter.
2.- During execution, when inside the program a call has been to a subroutine (MCALL, PCALL) greater than 9999.

SOLUTION The solution for each cause is:
1.- The parameter only admits values between 0 and 9999.
2.- The subroutine number must be between 1 and 9999.

0142 'Values 0 thru 32767.'

DETECTED While editing machine parameters

CAUSE An attempt has been made to assign the wrong value to a parameter.

SOLUTION The parameter only admits values between 0 and 32767.

0144 'Values 0 thru 65535.'

DETECTED While editing machine parameters

CAUSE An attempt has been made to assign the wrong value to a parameter.

SOLUTION The parameter only admits values between 0 and 65535.

0145 'Format +/- 5.5.'

DETECTED While editing machine parameters

CAUSE An attempt has been made to assign the wrong value to a parameter.

SOLUTION The parameter only admits values with the format: ± 5.5.

0147 'Numerical format exceeded.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE A data or parameter has been assigned a value greater than the established format.

SOLUTION Correct the syntax of the block. Most of the time, the numeric format will be 5.4 (5 integers and 4 decimals).

0148 'Text too long.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE While programming in high level language, the "ERROR" or "MSG" instruction has been assigned a text with more than 59 characters.

SOLUTION Correct the syntax of the block. The "ERROR" and "MSG" instructions cannot be assigned texts longer than 59 characters.

0149 'Incorrect message.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE While programming in high level language, the text associated with the "ERROR" or "MSG" instruction has been edited wrong.

SOLUTION Correct the syntax of the block. The programming format is:
(MSG "message")
(ERROR number, "message")
The message must be between " ".

0150 'Incorrect number of bits.'

DETECTED While editing tables.

CAUSE/S The various causes might be:
1.- In the «M» function table, in the section on customizing bits:
- The number does not have 8 bits.
- The number does not consist of 0's and 1's.

2.- In the machine parameter table, an attempt has been made to assign the wrong value of bit to a parameter.

SOLUTION The solution for each cause is:
1.- The customizing bits must consist of 8 digits of 0's and 1's.
2.- The parameter only admits 8-bit or 16-bit numbers.

0152 'Incorrect parametric programming.'

DETECTED During execution.

CAUSE The parameter has a value that is incompatible with the function it has been assigned to.

SOLUTION This parameter may have taken the wrong value, in the program history. Correct the program so this parameter does not reach the function with that value.

0154 'Insufficient memory.'

DETECTED During execution.

CAUSE The CNC does not have enough memory to internally calculate the paths.

SOLUTION Sometimes, this error is taken care of by changing the machining conditions.

0156 'Don't program G33 ,G95 or M19 S with no spindle encoder'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE A "G33", "G95" or "M19 S" has been programmed without having an encoder on the spindle.

SOLUTION If the spindle does not have an encoder, functions "M19 S", "G33" or "G95". Spindle machine parameter "NPULSES (P13)" indicates the number of encoder pulses per turn.

0157 'G79 not allowed when there is no active canned cycle.'

DETECTED During execution.

CAUSE An attempt has been made to execute the «Modification of canned cycle parameters (G79)» function without any canned cycle being active.

SOLUTION The "G79" function modifies the values of a canned cycle; therefore, there must be an active canned cycle and the "G79" must be programmed in the influence zone of that canned cycle.

0158 'Tool T must be programmed with G67 and G68.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE In the «Irregular pocket canned cycle with islands (G66)» the tool has not been defined for roughing "G67" (2D/3D pockets) for semi-finishing "G67" (3D pocket) or finishing "G68" (2D/3D pocket).

SOLUTION The irregular pocket canned cycle with islands requires the programming of the roughing tool "G67" (2D/3D pockets), the semi-finishing tool "G67" (3D pocket) and the finishing tool "G68" (2D/3D pocket).

0159 'Inch programming limit exceeded.'

DETECTED During execution.

CAUSE An attempt has been made to execute in inches a program edited in millimeters.

SOLUTION Enter function G70 (inch programming) or G71 (mm programming) at the beginning of the program.

0161 'G66 must be programmed before G67 and G68.'

DETECTED During execution.

CAUSE A roughing operation "G67" (2D/3D pockets), a semi-finishing operation "G67" (3D pocket) or a finishing operation "G68" (2D/3D pocket) has been programmed without having previous programmed the call to an «Irregular pocket canned cycle with islands (G66)».

SOLUTION When working with irregular pockets, before programming the aforementioned cycles, the call to the «Irregular canned cycle with islands (G66)» must be programmed.

0162 'No negative radius allowed with absolute coordinates'

DETECTED During execution.

CAUSE While operating with absolute polar coordinates, a movement with a negative radius has been programmed.

SOLUTION Negative radius cannot be programmed when using absolute polar coordinates.

0163 'The programmed axis is not longitudinal.'

DETECTED During execution.

CAUSE An attempt has been made to modify the coordinates of the point where the canned cycle is to be executed using the «Modification of the canned cycle parameters (G79)»function.

SOLUTION With "G79", the parameters defining a canned cycle may be modified, except the coordinates of the point where it will be executed. To change those coordinates, program only the new coordinates.

0164 'Wrong password.'

DETECTED While assigning protections.

CAUSE [ENTER] has been pressed before selecting the type of code to be assigned a password.

SOLUTION Use the softkeys to select the type of code to which a password is to be assigned.

0165 'Password: use uppercase/lowercase letters or digits.'

DETECTED While assigning protections.

CAUSE A bad character has been entered in the password.

SOLUTION The password can only consist of letters (upper and lower case) or digits.

0166 'Only one HIRTH axis per block is allowed.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE A movement has been programmed which involves the movement of two HIRTH axes simultaneously.

SOLUTION Only one HIRTH axis can be moved at a time.

0167 'Position-only rotary axis: Absolute values 0 - 359.9999'

DETECTED During execution.

CAUSE A movement of a positioning-only rotary axis has been programmed. The movement has been programmed in absolute coordinates (G90) and the target coordinate of the movement is not within the 0 to 359.9999 range.

SOLUTION Positioning-only rotary axes: In absolute coordinates, only movements within the 0 to 359.9999 range are possible.

0168 'Rotary axis: Absolute values (G90) within +/-359.9999.'

DETECTED During execution.

CAUSE A movement of a rotary axis has been programmed. The movement has been programmed in absolute coordinates (G90) and the target coordinate of the movement is not within the 0 to 359.9999 range.

SOLUTION Rotary axes: In absolute coordinates, only movements within the 0 to 359.9999 range are possible.

0169 'Modal subroutines cannot be programmed.'

DETECTED While executing in MDI mode

CAUSE An attempt has been made to call upon a modal subroutine (MCALL).

SOLUTION MCALL modal subroutines cannot be executed from the menu option «MDI execution».

0171 'The window must be previously defined.'

DETECTED During normal execution or execution through the user channel.

CAUSE An attempt has been made to write in a window (DW) that has not been previously defined (ODW).

SOLUTION It is not possible to write in a window that has not been previously defined. Check that the window to write in (DW) has been previously defined.

0172 'The program is not accessible'

DETECTED During execution.

CAUSE An attempt has been made to execute a program that cannot be executed.

SOLUTION The program may be protected against execution. To know if the program can be executed, check the attributes column, if the letter «X» is missing, it means that it cannot be executed.

0174 'Circular (helical) interpolation not possible.'

DETECTED During execution.

CAUSE An attempt has been made to execute a helical interpolation while the «LOOK-AHEAD (G51)» function was active.

SOLUTION Helical interpolations are not possible while the «LOOK-AHEAD (G51)» function is active.

0175 'Analog inputs: ANAI(1-8) = +/-5 Volts.'

DETECTED During execution.

CAUSE An analog input has taken a value out of the ±5V range.

SOLUTION Analog inputs may only take values within the ±5V range.

0176 'Analog outputs: ANAO(1-8) = +/-10 Volts.'

DETECTED During execution.

CAUSE An analog output has been assigned a value out of the ±10V range.

SOLUTION Analog outputs may only take values within the ±10V range.

0178 'G96 only possible with analog spindle.'

DETECTED During execution.

CAUSE The "G96" function has been programmed but either the spindle speed is not controlled or the spindle does not have an encoder.

SOLUTION To operate with the "G96" function, the spindle speed must be controlled (SPDLTYPE(P0)=0) and the spindle must have an encoder (NPULSES(P13) other than zero).

0180 'Program DNC1/2, HD or CARD A (optional).'

DETECTED While editing or executing.

CAUSE While programming in high level language, in the "OPEN" and "EXEC" instructions, an attempt has been made to program a parameter other than DNC1/2, HD or CARD A, or the DNC parameter has been assigned a value other than 1 or 2.

SOLUTION Check the syntax of the block.

0181 'Program A (append) or D (delete).'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE In the "OPEN" instruction the A/D parameter is missing.

SOLUTION Check the syntax of the block. The programming format is:

(OPEN P———,A/D,…)

Where:

- A : Appends new blocks after the existing ones.
- D : Deletes the existing program and it opens it as a new one.

0182 'Option not available.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE A «G» function has been defined which is not a software option.

0183 'Cycle does not exist.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE In the "DIGIT" instruction, a digitizing cycle has been defined which is not available.

SOLUTION The "DIGIT" instruction only admits two types of digitizing:

(DIGIT 1,…) : Grid pattern digitizing cycle.

(DIGIT 2,…) : Arc pattern digitizing cycle.

0185 'Tool offset does not exist'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE Within the block syntax, a tool offset has been called upon which is greater than the ones allowed by the manufacturer.

SOLUTION Program a new smaller tool offset.

0188 'Function not possible from PLC.'

DETECTED During execution.

CAUSE From the PLC channel and using the "CNCEX" instruction, an attempt has been made to execute a function that is incompatible with the PLC channel execution.

SOLUTION The installation manual (chapter 11.1.2) offers a list of the functions and instructions that may be executed through the PLC channel.

0190 'Programming not allowed while in tracing mode.'

DETECTED During execution.

CAUSE Among the blocks defining the «Tracing and digitizing canned cycles (TRACE)», there is block that contains a «G» function which does not belong in the profile definition.

SOLUTION The «G» functions available in the profile definition are:

G00	G01	G02	G03	G06	G08
G09	G36	G39	G53	G70	G71
G90	G91	G93			

0191 'Do not program tracing axes.'

DETECTED During execution.

CAUSE An attempt has been made to move an axis that has been defined as a tracing axis using the "G23" function.

SOLUTION The tracing axes are controlled by the CNC. To deactivate the tracing axes, use the "G25" function..

0192 'Incorrect active plane and longitudinal axis.'

DETECTED During execution.

CAUSE While programming in high level language, an attempt has been made to execute a probing cycle using the "PROBE" instruction, but the longitudinal axis is included in the active plane.

SOLUTION The "PROBE" probing canned cycles are executed on the X, Y, Z axes, the active plane being formed by two of them. The other axis must be perpendicular and it must be selected as the longitudinal axis.

0193 'G23 has not been programmed.'

DETECTED During execution.

CAUSE Digitizing "G24" has been activated or a tracing contour "G27" has been programmed, but without previously activating the tracing function "G23".

SOLUTION To digitize or operate with a contour, the tracing function must be activated previously.

0194 'Repositioning not allowed.'

DETECTED During execution.

CAUSE The axes cannot be repositioned using the "REPOS" instruction because the subroutine has not been activated with one of the interruption inputs.

SOLUTION Before executing the "REPOS" instruction, one of the interruption inputs must be activated.

0195 'Axes X, Y or Z slaved or synchronized.'

DETECTED During execution.

CAUSE While programming in high level language, an attempt has been made to execute a probing cycle using the "PROBE" instruction, but one of the X, Y or Z axis is slaved or synchronized.

SOLUTION To execute the "PROBE"¨ instruction, the X, Y, Z axes must not be slaved or synchronized. To unslave the axes, program "G78".

0196 'Axes X, Y and Z must exist.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE While programming in high level language, an attempt has been made to edit the "PROBE" instruction, but one of the X, Y or Z axis is missing.

SOLUTION To operate with the "PROBE" instruction, the X, Y, Z axes must be defined.

0198 'Deflection out of range.'

DETECTED During execution.

CAUSE In the tracing cycle "G23", a nominal probe deflection has been defined which is greater than the value set by machine parameter.

SOLUTION Program a smaller nominal probe deflection.

0199 'Preset of rotary axes: Values between 0-359.9999. '

DETECTED While presetting coordinates.

CAUSE An attempt has been made preset the coordinates of a rotary axis with a value out of the 0 to 359.9999 range.

SOLUTION The preset value of rotary axes must be within the 0 to 359.9999 range.

0200 'Program: G52 axis +/-5.5.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE When programming the «Movement against a hard stop (G52)», either the axis to be moved has not been programmed or several axes have been programmed.

SOLUTION When programming the "G52" function, the axis to be moved must be programmed but only one axis at a time.

0202 'Program G27 only when tracing a profile.'

DETECTED During execution.

CAUSE A tracing contour (G27) has been defined, but the tracing function is neither bi-dimensional nor three-dimensional.

SOLUTION The «Definition of a tracing contour (G27)» function must only be defined when tracing or digitizing in two or three dimensions.

0204 'Incorrect tracing method.'

DETECTED During execution.

CAUSE While executing a manual tracing "G23", an attempt has been made to jog a «follower» axis with the jog keys or the electronic handwheels.

SOLUTION When executing a manual tracing, the axes selected as followers are moved by hand. The rest may be jogged with the jog keys or the electronic handwheels.

0205 'Incorrect digitizing method.'

DETECTED During execution.

CAUSE Point-to-point digitizing has been defined, but the CNC is not in jog mode (it is in either in simulation or execution mode, instead).

SOLUTION To execute point-to-point digitizing, the CNC must be in jog mode.

0206 'Values 0 thru 6.'

DETECTED While editing machine parameters

CAUSE An attempt has been made to assign the wrong value to a parameter.

SOLUTION The parameter only admits values between 0 and 6.

0207 'Complete Table.'

DETECTED While editing tables.

CAUSE In the tables for «M» functions or tool offsets, an attempt has been made to define more data than those allowed by the manufacturer by means of machine parameters. When loading a table via DNC, the CNC does not delete the previous table, it replaces the existing values and it copies the new data in the free positions of the table.

SOLUTION The maximum number of data that can be defined is limited by the machine parameters:

- Maximum number of «M» functions : NMISCFUN(P29).
- Maximum number of : NTOOL(P23).
- Maximum number of tool offset : NTOFFSET(P27).
- Maximum number of magazine positions : NPOCKET(P25).

To load a new table via DNC, the previous table should be deleted.

0208 'Program A from 0 to 255'

DETECTED During execution.

CAUSE In the «LOOK-AHEAD (G51)» function, parameter "A" (% of acceleration to be applied) has been programmed with a value greater than 255.

SOLUTION Parameter "A" is optional, but when programmed, it must have a value between 0 and 255.

0209 'Program nesting not allowed.'

DETECTED During execution.

CAUSE From a running program, an attempt has been made to execute another program with the "EXEC" instruction which in turn also has an "EXEC" instruction.

SOLUTION Another program cannot be called upon from a program being executed using the "EXEC" instruction.

0210 'No compensation is permitted.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE An attempt has been made to activate or cancel tool radius compensation (G41, G42, G40) in a block containing a nonlinear movement.

SOLUTION Tool radius compensation must be activated/deactivated in linear movements (G00, G01).

0211 'Do not program a zero offset without cancelling the previous one.'

DETECTED During execution.

CAUSE An attempt has been made to define an incline plane using the «Definition of the incline plane (G49)» function while another one was already defined.

SOLUTION To define a new incline plane, the one previously defined must be canceled first. To cancel an incline plane, program "G49" without parameters.

0212 'Programming not permitted while G47-G49 are active.'

DETECTED During execution.

CAUSE While programming in high level language, an attempt has been made to execute a probing cycle with the "PROBE" instruction while function "G48" or "G49" was active.

SOLUTION The digitizing cycles "PROBE" are carried out on the X, Y, Z axes. Therefore, neither the "G48" nor the "G49" function may be active when executing them.

0213 'For G28 or G29, a second spindle is required.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE An attempt has been made to select the work spindle with "G28/G29", but the machine only has one work spindle.

SOLUTION If the machine only has one work spindle, the "G28/ G29" functions cannot be programmed.

0214 'Invalid G function when selecting a profile'

DETECTED While restoring a profile.

CAUSE Within the group of blocks selected to restore the profile, there is a block containing a «G» code that does not belong in the profile definition.

SOLUTION The «G» functions available in the profile definition are:

G00	G01	G02	G03	G06	G08
G09	G36	G37	G38	G39	G90
G91	G93				

0215 'Invalid G function after first point of profile'

DETECTED While restoring a profile.

CAUSE Within the selected blocks for restoring the profile, and after the starting point of a profile, there is a block containing a «G» function that does not belong in the profile definition.

SOLUTION The «G» functions available in the profile definition are:

G00	G01	G02	G03	G06	G08
G09	G36	G37	G38	G39	G90
G91	G93				

0216 'Nonparametric assignment after first point of profile'

DETECTED While restoring a profile.

CAUSE Within the selected blocks for restoring the profile, and after the starting point of a profile, a nonparametric assignment has been programmed in high level language (a local or global parameter).

SOLUTION The only high level instructions that can be edited are assignments to local parameters (P0 thru P25) and global parameters (P100 thru P299).

0217 'Invalid programming after first point of profile'

DETECTED While restoring a profile.

CAUSE Within the selected blocks for restoring the profile, and after the starting point of a profile, there is a high level block that is not an assignment.

SOLUTION The only high level instructions that can be edited are assignments to local parameters (P0 thru P25) and global parameters (P100 thru P299).

0218 'The axis cannot be programmed after first point of profile'

DETECTED While restoring a profile.

CAUSE Within the selected blocks for restoring the profile, and after the starting point of a profile, a position has been defined on an axis that does not belong to the active plane. A surface coordinate may have been defined after the starting point of the profile.

SOLUTION The surface coordinate of the profiles is only defined in the starting block of the first profile, the one corresponding to the starting point of the outside profile.

0219 'First point programmed wrong when selecting profile'

DETECTED While selecting a profile.

CAUSE The starting point of the profile has been programmed wrong. One of the two coordinates defining its position is missing.

SOLUTION The starting point of a profile must be defined on the two axes forming the active plane.

0226 'A tool cannot be programmed with G48 active'

DETECTED During execution.

CAUSE A tool change has been programmed while the «TCP transformation (G48)» function is active.

SOLUTION A tool change cannot take place while TCP transformation is active. To make a tool change, cancel TCP transformation first.

0227 'Program Q between +/-359.9999.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE In the «Electronic threading (G33)» function, the entry angle "Q" has been programmed with a value out of the ±359.9999 range.

SOLUTION Program an entry angle within the ±359.9999 range.

0228 'Do not program "Q" with parameter M19TYPE=0.'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE In the «Electronic threading (G33)» function, an entry angle "Q" has been programmed, but the type of spindle orientation available does not allow this operation.

SOLUTION In order to define an entry angle, spindle machine parameter M19TYPE(P43) must be set to «1».

0229 'Program maximum X'
0230 'Program minimum Y'
0231 'Program maximum Y'
0232 'Program minimum Z'
0233 'Program maximum Z'

DETECTED While editing at the CNC or while executing a program transmitted via DNC.

CAUSE While programming in high level language, in the "DGWZ" instruction, the indicated limit is missing or it has been defined with a non-numerical value.

SOLUTION Check the syntax of the block.

0234 'Wrong graphic limits'

DETECTED During execution.

CAUSE One of the lower limits defined with the "DGWZ" instruction is greater than its corresponding upper limit.

SOLUTION Program the upper limit of the graphics display area greater than the lower ones.

PREPARATION AND EXECUTION ERRORS

1000 'Not enough information about the path'

DETECTED During execution.

CAUSE The program has too many consecutive blocks without path data to apply tool radius compensation, rounding, chamfers or tangential entry / exit.

SOLUTION In order to carry out these operations, the CNC needs to know the path in advance; therefore, there cannot be more than 48 consecutive blocks without the path to be followed.

1001 'Plane change during rounding or chamfering'

DETECTED During execution.

CAUSE A plane change has been programmed on the path following a «Controlled corner rounding (G36)» or a «Chamfer (G39)».

SOLUTION The plane cannot be changed while executing a rounding or a chamfer. The path following the definition of a corner rounding or chamfer must be in the same plane as the rounding or chamfer.

1002 'Rounding radius too large '

DETECTED During execution.

CAUSE In the «Controlled corner rounding (G36)» function, a rounding radius has been programmed larger than one of the paths where it is defined.

SOLUTION The rounding radius must be smaller than the paths defining it.

1003 'Rounding in last block'

DETECTED During execution.

CAUSE A «Controlled corner rounding (G36)» or a «Chamfer (G39)» has been defined on the last path of the program or when the CNC cannot find information about the path following the definition of the corner rounding or chamfer.

SOLUTION A corner rounding or chamfer must be defined between two paths.

1004 'Tangential exit programmed incorrectly'

DETECTED During execution.

CAUSE The movement following a tangential exit (G38) is a circular path.

SOLUTION The movement following a tangential exit (G38) must be straight line.

1005 'Chamfer programmed incorrectly'

DETECTED During execution.

CAUSE The movement following a chamfer (G39) is a circular path.

SOLUTION The movement following a chamfer (G39) must be a straight line.

1006 'Chamfer value too large'

DETECTED During execution.

CAUSE In the «Chamfer (G39)» function, a chamfer has been programmed larger than the paths where it has been defined.

SOLUTION The chamfer must be smaller than the paths defining it.

1007 'G8 defined incorrectly'

DETECTED During execution.

CAUSE/S The probable causes might be:
1.- When programming a full circle with the «Arc tangent to previous path (G08)» function.
2.- When the tangent path ends at one point of the previous path or on its extension (In straight line).
3.- While operating with an irregular pocket with islands, when programming a "G08" function in the block following the definition of the beginning of the profile (G00).

SOLUTION The solution for each cause is:
1.- Full circles cannot be programmed using function "G08".
2.- The tangent path cannot end at one point of the previous path or on its extension (In straight line).
3.- The CNC does not have information about the previous path and it cannot execute a tangent arc.

1008 'There is no information on previous path'

DETECTED During execution.

CAUSE An arc tangent to the previous path has been programmed with "G08", but there isn't enough information about the previous path.

SOLUTION In order to make a path tangent to a previous one, there must be information about the previous path and it must be in the 48 blocks prior to the tangent path.

1009 'There is no information for arctangent in irregular pocket'

DETECTED During execution.

CAUSE Within the set of blocks defining a profile of an irregular pocket with islands, an arc tangent has been programmed, but some data is missing or there is not enough information on the previous path.

SOLUTION Check the profile defining data.

1010 'Wrong plane in tangential path'

DETECTED During execution.

CAUSE A plane change has been programmed between the definition of the «Arc tangent to previous path (G08)» function and the previous path.

SOLUTION The plane change cannot be done between both paths.

1011 'Jog movement out of limits'

DETECTED During execution.

CAUSE After defining an incline plane, the tool stays positioned at a point out of the work limits and an attempt has been made to jog an axis that does not position the tool inside the area defined by the work limits.

SOLUTION Jog the axis that allows positioning the tool inside the work limits.

1012 'G48 cannot be programmed with G43 active'

DETECTED During execution.

CAUSE An attempt has been made to activate TCP transformation (G48) while tool length compensation (G43) is active.

SOLUTION To activate TCP transformation (G48), tool length compensation must be canceled because TCP itself already implies a specific tool length compensation.

1013 'G43 cannot be programmed with G48 active'

DETECTED During execution.

CAUSE An attempt has been made to activate tool length compensation (G43) while TCP transformation (G48) is active.

SOLUTION Tool length compensation (G43) cannot be activated while TCP transformation (G48) is active because TCP itself already implies a specific tool length compensation.

1015 'Tool not defined in tool table'

DETECTED During execution.

CAUSE A tool change has been defined, but the new tool is not defined in the tool table.

SOLUTION Define the new tool in the tool table.

1016 'The tool is not in the tool magazine'

DETECTED During execution.

CAUSE A tool change has been defined, but the new tool is not defined in any table position of the tool magazine.

SOLUTION Define the new tool in the tool magazine table.

1017 'There is no empty pocket in the tool magazine'

DETECTED During execution.

CAUSE A tool change has been defined, but there isn't any pockets in the magazine to place the tool that currently is in the spindle.

SOLUTION The new tool may be defined in the tool table as special and more than magazine position may be reserved for it. In that case, that position is fixed for that tool and it cannot be occupied by another tool. To avoid this error message, a free position should be left in the tool magazine.

1018 'A tool change has been programmed without M06'

DETECTED During execution.

CAUSE After searching for a tool and before searching for the next one, an M06 has not been programmed.

SOLUTION This error comes up when having a machining center (general machine parameter TOFFM06(P28)=YES) which has a cyclic automatic tool changer (general machine parameter CYCATC(P61)=YES). In that case, after searching for a tool and before searching for the next one, a tool change has to be made using an M06.

1019 'There is no tool of the same family to replace it'

DETECTED During execution.

CAUSE The real life of the requested tool exceeds its nominal life. The CNC has tried to replace it with another one of the same family (type), it has found none.

SOLUTION Replace the tool or define another one of the same family.

1020 'Do not use high level to change active tool or next one'

DETECTED During execution.

CAUSE While programming in high level language using the "TMZT" variable, an attempt has been made to assign the active tool (or the next one) to a magazine position.

SOLUTION To change the active tool or the next one, use the «T» function. The active tool or the next one cannot be moved to the magazine using the "TMZT" variable.

1021 'The canned cycle is missing a tool offset'

DETECTED During execution.

CAUSE A probing canned cycle "PROBE" has been programmed for tool calibration, but no tool offset has been selected.

SOLUTION To execute the «Tool calibration canned cycle (PROBE)», the tool offset that is supposed to store the data of the probing cycle must be previously selected.

1023 'G67. Tool radius too large'

DETECTED During execution.

CAUSE In the «Irregular pocket canned cycle with islands (G66), a tool has been selected with too large of a radius for the roughing operation "G67" (2D pocket). The tool does not fit in the pocket.

SOLUTION Select a smaller tool radius.

1024 'G68. Tool radius too large'

DETECTED During execution.

CAUSE In the «Irregular pocket canned cycle with islands (G66)», a tool has been selected with too large of a radius for the finishing operation "G68" (2D pocket). Somewhere in the machining operation, the distance between the outside profile and the profile of an island is smaller than the tool diameter.

SOLUTION Select a smaller tool radius.

1025 'A tool with no radius has been programmed'

DETECTED During execution.

CAUSE In the «Irregular pocket canned cycle with islands (G66)», an operation has been programmed (G67/G68) with a tool having a «0» radius.

SOLUTION Correct the tool geometry in tool table, or select another tool for that operation.

1026 'A step greater than the tool diameter has been programmed'

DETECTED During execution.

CAUSE In the «Rectangular pocket canned cycle (G87)», in the «Circular pocket canned cycle (G88)» or in a «Irregular pocket canned cycle with islands (G66)», parameter "C" has been programmed with a value larger than the tool diameter being used for that operation.

SOLUTION Correct the block syntax. The machining step "C" must be smaller than or equal to the tool diameter.

1027 'A tool cannot be programmed with G48 active'

DETECTED During execution.

CAUSE A tool change has been programmed while «TCP transformation (G48)» is active.

SOLUTION A tool change is not possible while TCP transformation is active. TCP transformation must be canceled before making tool change.

1028 'Do not switch axes over or back while G15, G23, G48 or G49 are active'

DETECTED During execution.

CAUSE An attempt has been made to switch an axis or switch it back (G28/G29) while the "G15", "G23", "G48" or "G49" function was active.

SOLUTION The axes cannot be switched while the "G15", "G23", "G48", "G49" are active.

1029 'Do not switch axes already switched over'

DETECTED During execution.

CAUSE An attempt has been made to switch an axis (G28) which is already switched with another one.

SOLUTION An axis switched with another one cannot be directly switched with a third one. It must be switched back first. (G29 axis).

1030 'Not enough room for the automatic range change M code'

DETECTED During execution.

CAUSE While using an automatic gear change and having programmed in a block seven «M» functions and an «S» function involving a tool change, the CNC cannot include the «M» for the automatic tool change in that block.

SOLUTION Program one of the «M» functions or the «S» function in a separate block.

1031 'A subroutine is not allowed for automatic range change'

DETECTED During execution.

CAUSE In machines using an automatic gear change, when programming an «S» speed that involves a gear change and the «M» function for the automatic gear change has a subroutine associated with it.

SOLUTION When using an automatic gear change, the «M» functions for the gear change cannot have an associated subroutine.

1032 'Spindle speed range not defined for M19'

DETECTED During execution.

CAUSE An "M19" has been programmed, but none of the gear change functions is active ("M41", "M42", "M43" or "M44").

SOLUTION On power-up, the CNC does not assume any gear. Therefore, if the gear change function is not automatically generated (spindle parameter AUTOGEAR(P6)=NO), the auxiliary functions must be programmed for the gear change ("M41", "M42", "M43" or "M44").

1033 'Incorrect range change'

DETECTED During execution.

CAUSE/S The various probable causes are:

1.- When trying to make a gear change and the machine parameters for the gears (MAXGEAR1, MAXGEAR2, MAXGEAR3, or MAXGEAR4) are set wrong. All the gears have not be used and the unused ones have been set to maximum speed of zero.

2.- When a gear change has been programmed ("M41", "M42", "M43" or "M44"), but the PLC has not responded with corresponding active gear signal (GEAR1, GEAR2, GEAR3 or GEAR4).

SOLUTION The solution for each cause is:

1.- When not using all four gears, the lowest ones must be used starting with "MAXGEAR1", and the unused gears must be assigned the highest value of the ones used.

2.- Check the PLC program.

1034 'S has been programmed without an active range'

DETECTED During execution.

CAUSE An attempt has been made to start the spindle, but no gear has been selected.

SOLUTION On power-up, the CNC does not assume any gear. Therefore, if the gear change function is not automatically generated (spindle parameter AUTOGEAR(P6)=NO), the auxiliary functions must be programmed for the gear change ("M41", "M42", "M43" or "M44").

1035 'S programmed too large'

DETECTED During execution.

CAUSE An «S» value has been programmed that is greater than the maximum value allowed for the last active range (gear).

SOLUTION Program a smaller «S» value.

1036 'S not programmed in G95 or threadcutting'

DETECTED During execution.

CAUSE Either the feedrate has been programmed in mm (inches) per rev. (G95) or the «Electronic threading (G33)» without having a spindle speed selected.

SOLUTION Working in mm/rev. (G95) or making an thread (using G33) requires the programming of an "S" speed.

1040 'Canned cycle does not exist'

DETECTED During execution in MDI mode.

CAUSE An attempt has been made to execute a canned cycle (G8x) after interrupting a program while executing a canned cycle (G8x) and then doing a plane change.

SOLUTION Do not interrupt the program while executing a canned cycle.

1041 'A parameter required by the canned cycle has not been programmed'

DETECTED During execution.

CAUSE/S The probable causes might be:

1.- Some parameter is missing in the «irregular pocket canned cycle with islands».

2D POCKETS

- In the roughing operation "G67", either parameter "I" or "R" has not been programmed.

3D POCKETS

- In the roughing operation "G67", either parameter "I" or "R" has not been programmed.
- There is no roughing operation and the semifinishing operation "G67" has either the "I" or "R" parameter missing.
- There is no roughing operation and the finishing operation "G68" has either the "I" or "R" parameter missing.
- The finishing operation "G68" has the "B" parameter missing.

2.- The digitizing canned cycle has some parameter missing.

SOLUTION Correct the parameter definition.

Pocket with islands (finishing operation)

This cycle requires the programming of parameters "I" and "R" in the roughing operation. If there is no roughing operation, they must be defined in the finishing operation (2D) or in the semifinishing operation (3D). If there is no semifinishing operation (3D), they must be defined in the finishing operation. In 3D pockets, parameter "B" must be programmed in the finishing operation.

Digitizing cycles

Check the syntax of the block. The programming formats are:

(DIGIT 1,X,Y,Z,I,J,K,B,C,D,F)
(DIGIT 2,X,Y,Z,I,J,K,A,B,C,F)

1042 'Invalid parameter value in canned cycle'

DETECTED During execution.

CAUSE/S The probable causes might be:

1.- In the «Irregular pocket canned cycle with islands» when defining a parameter with the wrong value in the finishing operation "G68". Maybe it has been assigned a negative (or zero) value when it only admits positive values.

2.- In the «Irregular pocket canned cycle with islands», when parameter "B", "C" or "H" has been defined with zero value in the drilling operation (G69).

3.- In the rectangular (G87) or circular (G88) pocket canned cycles, either parameter "C" or a pocket dimension has been defined with zero value.

4.- In the «Deep hole drilling canned cycle with variable peck (G69)», parameter "C" has been defined with zero value.

5.- In the digitizing canned cycle, some parameter has been assigned the wrong value. Maybe it has been assigned a negative (or zero) value when it only admits positive values.

SOLUTION Correct the parameter definition.

Pocket with islands (finishing operation)

Parameter "Q" only admits values: 0, 1 or 2.
Parameter "B" only admits values other than zero.
Parameter "J" must be smaller than the tool radius used for that operation

Grid pattern digitizing.

Parameter "B" only admits positive values greater than zero.
Parameter "C" only admits positive values greater than zero.
Parameter "D" only admits values: 0 or 1.

Arc pattern digitizing

Parameter "J" and "C" only admit positive values greater than zero.
Parameter "K", "A" and "B" only admits positive values.

1043 'Wrong depth-profile in irregular pocket'

DETECTED During execution.

CAUSE In «Irregular pocket canned cycle with islands» (3D):

- The depth profiles of two sections of the same contour (simple or composite) intersect.
- The finishing of a contour cannot be done with the programmed tool (spheric path with a non-spheric tool).

SOLUTION The depth profiles of two sections of the same profile cannot intersect. On the other hand, the depth profile must be defined after the plane profile and the same starting point must be used on both profiles. Check that the selected tool tip is the most appropriate for the programmed depth profile.

1044 'Self-intersecting plane-profile in irregular pocket'

DETECTED During execution.

CAUSE In the profiles set defining a pocket with islands, there is a profile that intersects itself.

SOLUTION Check the profile definition. The profile of a pocket with islands cannot intersect itself.

1045 'Error when programming drilling an irregular pocket'

DETECTED During execution.

CAUSE In the «Irregular pocket canned cycle with islands (G66)», a canned cycle has been defined which is not a drilling canned cycle.

SOLUTION In the drilling operation, only "G81", "G82", "G83" or "G69" may be defined.

1046 'Wrong tool position prior to canned cycle'

DETECTED During execution.

CAUSE When calling a canned cycle, the tool is positioned between the reference plane and the final depth coordinate (bottom) of some operation.

SOLUTION When calling a canned cycle, the tool must be positioned above the reference plane.

1047 'Plane profile open in irregular pocket'

DETECTED During execution.

CAUSE In the profile set defining a pocket with islands, there is a profile that doesn't start and finish at the same point.

SOLUTION Check the profile definition. The profiles defining the pockets with islands must be closed. This error may come up because "G01" has not been programmed after starting a profile with "G00".

1048 'Part surface coordinate not programmed in irregular pocket'

DETECTED During execution.

CAUSE The surface coordinate of the pocket has not been programmed at the first point of the geometry definition.

SOLUTION The data corresponding to the surface coordinate must be defined in the first block defining the pocket profile (in absolute coordinates).

1049 'Wrong reference plane coordinate in canned cycle'

DETECTED During execution.

CAUSE In some operation of the «Irregular pocket canned cycle with islands (G66)», the coordinate of the reference plane is between the part surface coordinate and the final depth of some operation.

SOLUTION The reference plane must be above the part surface. Sometimes this error comes up as a result of programmed the part surface in incremental coordinates. The pocket surface data must be programmed in absolute coordinates.

1050 'Incorrect variable value'

DETECTED During execution.

CAUSE Too high a value has been assigned to a variable by means of parameters.

SOLUTION Check the program history, and make sure that that parameter does not reach the assignment block with that value.

1051 'Incorrect access to PLC variables'

DETECTED During execution.

CAUSE An attempt has been made to read a PLC variable from the CNC, but it was not defined in the PLC program.

1052 'Access to a variable with non-permitted index'

DETECTED While editing.

CAUSE While programming in high level language, an operation is carried out with either a local parameter greater than 25 or with a global parameter greater than 299.

SOLUTION The CNC uses the following parameters:
- Local: 0-25.
- Global: 100-299.

No other parameters can be used in the operations.

1053 'Local parameters not accessible'

DETECTED During execution in the user channel.

CAUSE An attempt has been made to execute a block containing an operation with local parameters.

SOLUTION The program executed in the user channel cannot carry out operations with local parameters (P0 through P25).

1054 'Local parameters not accessible'

DETECTED During execution.

CAUSE While programming in high level language, more than 6 nesting levels have been used with the "PCALL" statement within the same loop.

SOLUTION No more than 6 nesting levels are possible with local parameters within the 15 nesting levels for subroutines. Every time a call is made with the "PCALL" statement, a new nesting loop is generated for local parameters as well as for the subroutines.

1055 'Nesting exceeded.'

DETECTED During execution.

CAUSE While programming in high level language, more than 15 nesting levels have been used with the "CALL", "PCALL" or "MCALL" statements within the same loop.

SOLUTION No more than 15 nesting levels are possible. Every time a called is made with the "CALL", "PCALL" or "MCALL" statements, a new nesting level is generated.

1056 'RET not associated to a subroutine'

DETECTED During execution.

CAUSE The "RET" instruction has been edited without having previously edited the "SUB" instruction.

SOLUTION To use the "RET" instruction (end of subroutine), the subroutine must start with the "SUB" instruction (subroutine number).

1057 'Subroutine not defined'

DETECTED During execution.

CAUSE A call has been made (CALL, PCALL…) to a subroutine that is not defined in the CNC's memory.

SOLUTION Check that the name of the subroutine is correct and that it exists in the CNC's memory (not necessarily in the same program making the call).

1059 'Jump to an undefined label'

DETECTED During execution.

CAUSE While programming in high level language, the "GOTO N—" instruction has been programmed, but the programmed block number (N) does not exist.

SOLUTION When programming the "GOTO N—" instruction, the block it refers to must be defined in the same program.

1060 'Label not defined'

DETECTED During execution.

CAUSE/S The probable causes might be:

1.- While programming in high level language, the "RPTN—, N—" instruction has been programmed, but the programmed block number (N) does not exist.

2.- When in an «Irregular pocket canned cycle with islands (G66)» "G66 … S— E—" has been programmed, but one of the data defining the beginning or end of the profiles.

SOLUTION The solution for each cause is:

1.- When programming the "RPTN—, N—" instruction, the block it refers to must be defined in the same program.

2.- Check the program. Edit the label for the "S" parameter at the beginning of the profile definition and the label for the "E" parameter at the end of the profile definition.

1061 'Label cannot be searched'

DETECTED During execution in MDI mode

CAUSE While programming in high level language, an "RPT N—, N—" or "GOTO N—" instruction has been defined

SOLUTION "RPT" or "GOTO" type instructions cannot be programmed in MDI mode.

1062 'Subroutine not available in program'

DETECTED During execution.

CAUSE A subroutine has been called which is contained in a program that is currently being used by the DNC.

SOLUTION Wait for the DNC to be done with the program, If the subroutine is going to be used often, it is advisable to keep it in a separate program.

1063 'Program cannot be opened.'

DETECTED During execution.

CAUSE While running a program in infinite mode, an attempt has been made to execute another infinite program using the "EXEC" instruction at the running program.

SOLUTION Only one infinite program may be run at a time.

1064 'The program cannot be executed.'

DETECTED During execution.

CAUSE An attempt has been made to execute a program from another one using the "EXEC" instruction, but the program does not exit or is protected against execution.

SOLUTION The program to be executed with the "EXEC" instruction must be in CNC memory and it must be executable.

1065 'Beginning of compensation without a straight path'

DETECTED During execution.

CAUSE The first movement in the work plane after activating tool radius compensation (G41/G42) is not a linear movement.

SOLUTION The first movement after activating tool radius compensation (G41/G42) must be a linear movement.

1066 'End of compensation without a straight path'

DETECTED During execution.

CAUSE The first movement in the work plane after canceling tool radius compensation (G40) is not a linear movement.

SOLUTION The first movement after canceling tool radius compensation (G40) must be a linear movement.

1067 'Compensation radius too large'

DETECTED During execution.

CAUSE While working with tool radius compensation (G41/G42) an inside arc has been programmed with a radius smaller than the tool radius.

SOLUTION Use a tool with a smaller radius. When working with tool radius compensation (G41/G42), the arc radius must be greater than the tool radius. Otherwise, the tool cannot machine along the programmed path

1068 'Step in a straight path'

DETECTED During execution.

CAUSE While working with tool radius compensation (G41/G42), the profile has a straight section that cannot be machined because the tool diameter is too large.

SOLUTION Use a tool with a smaller radius.

1070 'Step in circular path'

DETECTED During execution.

CAUSE While working with tool radius compensation (G41/G42), the profile has a circular section that cannot be machined because the tool diameter is too large.

SOLUTION Use a tool with a smaller radius.

1071 'Compensation plane change'

DETECTED During execution.

CAUSE While working with tool radius compensation (G41/G42), another work plane has been selected.

SOLUTION To change the work plane, tool radius compensation must be canceled (G40).

1072 'Radius comp. not possible when positioning rotary axis'

DETECTED During execution.

CAUSE An attempt has been made to move a positioning-only rotary axis while tool radius compensation (G41/G42) is on.

SOLUTION Positioning-only rotary axes do not admit tool radius compensation. To cancel it, use the "G40" function.

1076 'Angle coordinate programmed incorrectly'

DETECTED During execution.

CAUSE While programming in the «angle-coordinate» format, an axis movement has been programmed with an angle perpendicular to that axis (v.g. the main plane is formed by the X, Y axes and the X axis is programmed to move at 90º).

SOLUTION Check and correct the definition of the movement in the program. When working with parameters, check that they reach the definition of the movement with the right values.

1077 'Arc programmed with radius too small or complete circle'

DETECTED During execution.

CAUSE/S The probable causes might be:

1.- When programming a full circle with the format: "G02/G03 X Y R".

2.- When programming with the format "G02/G03 X Y R", but the distance to the arc's end point is greater than the diameter of the programmed circle.

SOLUTION The solution for each cause is:

1.- With this format, full circles cannot be made. Program the end point with different coordinates from those of the starting point.

2.- The diameter of the circle must be greater than the distance to the arc's end point.

1078 'Negative radius in polar coordinates'

DETECTED During execution.

CAUSE While working in incremental polar coordinates, a block is executed which gives a negative final radius position.

SOLUTION When programming incremental polar coordinates, negative radius can be programmed, but the final (absolute) position of the radius must be positive.

1079 'There is no subroutine associated with G74'

DETECTED While executing a home search.

CAUSE/S The probable causes might be

1.- When trying to carry out a home search (on all axes) manually, but the associated subroutine indicating the searching sequence does not exist.

2.- Function "G74" has been programmed, but the associated subroutine indicating the searching sequence does not exist.

SOLUTION The solution for each cause is:

1.- To execute function "G74", its associated subroutine must be defined.

2.- If function "G74" is to be executed from a program, the home searching sequence for the axes may be defined.

1080 'Plane change during tool inspection'

DETECTED While executing the «tool inspection» option.

CAUSE The work plane has been changed, but it has not been restored before resuming execution.

SOLUTION Before resuming execution, the plane that was active before doing the «tool inspection» must be restored.

1081 'Block not allowed in MDI or during tool inspection'

DETECTED While executing the «tool inspection» option.

CAUSE An attempt has been made to execute the "RET" instruction.

SOLUTION This instruction cannot be executed within the «tool inspection» option.

1082 'Probe signal has not been received'

DETECTED During execution.

CAUSE/S The probable causes might be:

1.- A "PROBE" probing canned cycle has been programmed, but the probe has moved the maximum safety distance of the cycle without sending the probe signal to the CNC.

2.- When programming the "G75" function, the end coordinate has been reached without receiving the probe signal. (Only when general machine parameter PROBERR(P119)=YES).

SOLUTION The solution for each cause is:

1.- Check that the probe is connected properly.
The maximum probing distance (in the PROBE cycles) depends on the safety distance "B". To increase this distance, increase the safety distance.

2.- If PROBERR(P119)=NO, no error will be issued when this end coordinate is reached without receiving the probe signal (only with the "G75" function).

1083 'Range exceeded'

DETECTED During execution.

CAUSE The distance to travel for the axes very long and the programmed feedrate for that movement is very low.

SOLUTION Program a higher feedrate for this movement.

1084 'Circular path programmed incorrectly'

DETECTED During execution.

CAUSE/S The probable causes might be:

1.- When programming an arc using the format: "G02/G03 X Y I J", an arc cannot be made with the programmed radius and end point.

2.- When programming an arc using the format: "G09 X Y I J", The three points of the arc are in line or there are identical points.

3.- When trying to make a rounding or a tangential entry on a path not belonging to the active plane.

4.- When programming a tangential exit and the next path is tangent to (and on the linear extension of) the one prior to the tangential exit.

If the error comes up in the block calling the «Irregular pocket canned cycle with islands», it is because one of the aforementioned cases occurs in the set of blocks defining the profile of an irregular pocket with islands.

SOLUTION The solution for each cause is:

1.- Correct the syntax of the block. The coordinates of the end point or of the radius are defined wrong.

2.- The three points used to define the arc must be different and cannot be in line.

3.- Maybe a plane has been defined using "G16", "G17", "G18" or "G19". In that case, rounding, chamfers, and tangential entries/exits can be carried out on the main axes defining that plane. To make them in another plane, it must be selected before.

4.- The path after the tangential exit may be tangent, but it cannot be on the straight extension of the previous path.

1085 'Helical path programmed incorrectly'

DETECTED During execution.

CAUSE When programming an arc with the format: "G02/G03 X Y I J Z K" the programmed helical path cannot be carried out. The desired height cannot be reached with the programmed helical pitch.

SOLUTION Correct the syntax of the block. The height of the interpolation and the coordinates of the end point in the plane must be related taking the helical pitch into consideration.

1086 'The Spindle cannot be referenced (homed)'

CAUSE Spindle machine parameter REFEED1(P34) is set to «0».

1087 'Circle with zero radius'

DETECTED During execution.

CAUSE/S The probable causes might be:

1.- When programming an arc with the format: "G02/G03 X Y I J", a circular interpolation has been programmed with «zero» radius.

2.- While working with tool radius compensation, an inside arc has been programmed with a radius equal to the tool radius.

SOLUTION The solution for each cause is:

1.- Arcs with zero radius cannot be programmed. Program a radius value other than zero.

2.- When working with tool radius compensation, the arc radius must be greater than the tool radius. Otherwise, the tool cannot machine the programmed path because the tool would have to machine an arc with zero radius.

1088 'Zero offset range exceeded'

DETECTED During execution.

CAUSE A zero offset has been programmed and the end position has too high a value.

SOLUTION Check that the values assigned to the zero offsets (G54-G59) are correct. If the offset values have been assigned from a program using parameters, check that the parameter values are correct. If an absolute zero offset (G54-G57) has been programmed and an incremental one (G58-G59), check that the sum of both does not exceed the travel limits of the machine.

1089 'Work zone limit range exceeded'

DETECTED During execution.

CAUSE Work zone limits "G20" or "G21" have been programmed using parameters and the value of the parameter is greater than the one allowed for this function.

SOLUTION Check the program history so this parameter does not reach with that value to the block defining those limits.

1090 'Point within the forbidden zone 1'

DETECTED During execution.

CAUSE An attempt has been made to move an axis to a point located inside the work zone 1 which has been defined as «no entry zone».

SOLUTION In the history of the program, work zone 1 (defined with G20/G21) has been defined as «no entry zone» (G22 K1 S1). To disable it, program "G22 K1 S0".

1091 'Point within the forbidden zone 2'

DETECTED During execution.

CAUSE An attempt has been made to move an axis to a point located inside the work zone 2 which has been defined as «no entry zone».

SOLUTION In the history of the program, work zone 2 (defined with G20/G21) has been defined as «no entry zone» (G22 K2 S1). To disable it, program "G22 K2 S0".

1092 'Insufficient accelerations for the programmed threadcutting feedrate'

DETECTED During execution.

CAUSE A threading operation has been programmed with not enough room to accelerate and decelerate.

SOLUTION Program a lower feedrate.

1095 'Probe axes out of alignment.'

DETECTED During the probe calibration process.

CAUSE An axis has been moved touching the cube and some axis that has not moved registers a deflection greater than the value allowed by machine parameter MINDEFLE(P66). This is because the probing axes are not parallel enough to the axes of the machine.

SOLUTION Correct the parallelism between the probing axes and those of the machine.

1096 'Point within the forbidden zone 3'

DETECTED During execution.

CAUSE An attempt has been made to move an axis to a point located inside the work zone 3 which has been defined as «no entry zone».

SOLUTION In the history of the program, work zone 3 (defined with G20/G21) has been defined as «no entry zone» (G22 K3 S1). To disable it, program "G22 K3 S0".

1097 'Point within the forbidden zone 4'

DETECTED During execution.

CAUSE An attempt has been made to move an axis to a point located inside the work zone 4 which has been defined as «no entry zone».

SOLUTION In the history of the program, work zone 4 (defined with G20/G21) has been defined as «no entry zone» (G22 K4 S1). To disable it, program "G22 K4 S0".

1098 'Wrong work zone boundaries'

DETECTED During execution.

CAUSE The upper limits (G21) of the work zone defined are equal to or less than its lower limits (G20)

SOLUTION The upper limits (G21) of the work zone must always be greater than its lower limits (G20).

1099 'Do not program a slaved axis'

DETECTED During execution.

CAUSE While working with polar coordinates, a movement has been programmed which implies moving an axis which is slaved to another one.

SOLUTION The movements in polar coordinates are carried out on the main axes of the work plane. Therefore, the axes defining a plane cannot be slaved to each other or to a third axis. To free the axes, program "G78".

1100 'Spindle travel limit overrun'

DETECTED During execution.

CAUSE An attempt has been made to exceed the physical travel limits of the spindle. Consequently, the PLC activates the spindle marks: "LIMIT+S" or "LIMIT-S" ("LIMIT+S2" or "LIMIT-S2" when working with the second spindle)

1101 'Spindle locked'

DETECTED During execution.

CAUSE The CNC tries to output the analog voltage to the drive while the spindle input SERVOSON is still low. The error may come up due to an error in the PLC program where this signal is treated wrong or maybe the value of the spindle parameter DWELL(P17) is too low.

1102 'Spindle following error limit overrun'

DETECTED During execution.

CAUSE While the spindle is operating in closed loop (M19), its following error is greater than the values indicated by spindle parameters MAXFLWE1(P21) or MAXFLE2(P22). The probable causes for this error are:

DRIVE FAILURE
- Defective drive.
- Enable signals missing.
- Power supply missing.
- Poor drive adjustment.
- Velocity command signal missing.

MOTOR FAILURE
- Defective motor.
- Power wiring.

FEEDBACK FAILURE
- Defective feedback device.
- Defective feedback cable.

CNC FAILURE
- Defective CNC.
- Wrong parameter setting.

MECHANICAL FAILURE
- Mechanical friction.
- Spindle mechanically locked up

1110-1118 '* axis range exceeded'

DETECTED During execution.

CAUSE A movement has been defined using parameters and the value of the parameter is greater than the maximum axis travel allowed.

SOLUTION Check the history of the program so that parameter does not reach with that value to the block where that movement has been programmed.

1119-1127 '* axis cannot be synchronized'

DETECTED During execution.

CAUSE/S The probable causes might be:

1.- An attempt has been made to synchronize an axis with another one from the PLC, but the axis is already slaved to another one with function "G77".

2.- When programming or trying to move an axis already synchronized with another one.

1128-1136 '* axis maximum feed exceeded'

DETECTED During execution.

CAUSE The resulting feedrate of some axis after applying the particular scaling factor exceeds the maximum value indicated by axis machine parameter MAXFEED (P42).

1137-1145 'Incorrect * axis feedrate parameter'

DETECTED During execution.

CAUSE "G00" has been programmed with axis parameter G00FEED(P38)=0 or "G1 F00" has been programmed with axis machine parameter MAXFEED(P42) = 0.

1146-1154 '* axis locked'

DETECTED During execution.

CAUSE The CNC tries to output the velocity command to the drive while the spindle input SERVO(n)ON is still low. The error may come up due to an error in the PLC program where this signal is treated wrong or maybe the value of the spindle parameter DWELL(P17) is too low.

1155-1163 '* axis soft limit overrun'

DETECTED During execution.

CAUSE A coordinate has been programmed which is beyond the limits defined by axis machine parameters LIMIT+(P5) and LIMIT-(P6).

1164-1172 '* axis work zone 1 overrun'

DETECTED During execution.

CAUSE An attempt has been made to move an axis to a point located inside the work zone 1 which has been defined as «no exit zone».

SOLUTION In the history of the program, work zone 1 (defined with G20/G21) has been defined as «no exit zone» (G22 K1 S2). To disable it, program "G22 K1 S0".

1173-1181 '* axis work zone 2 overrun'

DETECTED During execution.

CAUSE An attempt has been made to move an axis to a point located inside the work zone 2 which has been defined as «no exit zone».

SOLUTION In the history of the program, work zone 2 (defined with G20/G21) has been defined as «no exit zone» (G22 K2 S2). To disable it, program "G22 K2 S0".

1182-1190 '* axis following error limit overrun'

DETECTED During execution.

CAUSE The following error of the axis is greater than the values indicated by spindle parameters MAXFLWE1(P21) or MAXFLE2(P22). The probable causes for this error are:

DRIVE FAILURE
- Defective drive.
- Enable signals missing.
- Power supply missing.
- Poor drive adjustment.
- Velocity command signal missing.

MOTOR FAILURE
- Defective motor.
- Power wiring.

FEEDBACK FAILURE
- Defective feedback device.
- Defective feedback cable.

CNC FAILURE
- Defective CNC.
- Wrong parameter setting.

MECHANICAL FAILURE
- Mechanical friction.
- Axis mechanically locked up

1191-1199 'Coupled * axis following error difference too large'

CAUSE The «n» axis is electronically coupled to another one or it is slaved to a Gantry axis and the difference between their following errors is greater than the value set by axis machine parameter MAXCOUPE(P45).

1200-1208 '* axis hard limit overrun'

DETECTED During execution.

CAUSE An attempt has been made to exceed the physical travel limits of the axis. Consequently, the PLC activates the axis marks: "LIMIT+1" or "LIMIT-1"

1209-1217 '* axis servo error'

CAUSE The actual axis speed, after a time period indicated by axis machine parameter FBALTIME(P12), is below 50% or over 200% of the programmed value.

1218-1226 '* axis work zone 3 overrun'

DETECTED During execution.

CAUSE An attempt has been made to move an axis to a point located inside the work zone 3 which has been defined as «no exit zone».

SOLUTION In the history of the program, work zone 3 (defined with G20/G21) has been defined as «no exit zone» (G22 K3 S2). To disable it, program "G22 K3 S0".

1227 'Wrong profile intersection in irregular pocket with islands'

DETECTED During execution.

CAUSE In the «Irregular pocket canned cycle with islands (G66)», there are two profiles in the plane which have the starting point or some section in common.

SOLUTION Define the profiles again. Two plane profiles cannot start at the same point or have sections in common.

1228-1236 '* axis work zone 4 overrun'

DETECTED During execution

CAUSE An attempt has been made to move an axis to a point located inside the work zone 4 which has been defined as «no exit zone».

SOLUTION In the history of the program, work zone 4 (defined with G20/G21) has been defined as «no exit zone» (G22 K4 S2). To disable it, program "G22 K4 S0".

1238 'Parameter range protected. Cannot be written. P297, P298'

DETECTED During execution

CAUSE An attempt has been made to execute the «definition of an incline plane (G49), but parameters P297 and P298 are write-protected with machine parameters ROPARMIN(P51) and ROPARMAX(P52).

SOLUTION During the definition of an incline plane, the CNC updates parameters P297 and P298. Therefore, these two parameters must not be write-protected.

HARDWARE ERRORS

2000 'External emergency activated'

DETECTED During execution.

CAUSE PLC input I1 has been set to zero (possible E-stop button) or the PLC mark M5000(/EMERGEN) has been set to zero.

SOLUTION Check at the PLC why these inputs are set to zero. (Maybe power is missing).

2001-2009 '* axis feedback error'

DETECTED During execution.

CAUSE The CNC does not receive feedback signals from the axes.

SOLUTION Check the feedback connections.

NOTE:This error comes up on differential feedback signals (double-ended signals), DIFFBACK(P9)=YES, and sinewave feedback signals SINMAGNI(P10) other than zero, when parameter FBACKAL(P11)=ON. This error can be avoided by setting parameter FBACKAL(P11)=OFF, although this solution is only temporary.

2010 'Spindle feedback error'

DETECTED During execution.

CAUSE The CNC does not receive the spindle feedback signals.

SOLUTION Check the feedback connections.

NOTE:This error comes up on differential feedback signals (double-ended signals), DIFFBACK(P14)=YES, when parameter FBACKAL(P15)=ON. This error can be avoided by setting parameter FBACKAL(P15)=OFF, although this solution is only temporary.

2011 'Maximum temperature exceeded'

DETECTED Any time.

CAUSE The maximum internal CNC temperature exceeded. The probable causes might be:

- Poor ventilation of the electrical cabinet (enclosure).
- Axis board with some defective component.

SOLUTION Turn the CNC off and wait until it cools off. If the error persists, some component of the board may be defective. In that case, contact the Service Department to replace the board.

2012 'Axes board without voltage'

DETECTED During execution.

CAUSE The 24V are missing from the outputs of the axes board. The fuse might be blown.

SOLUTION Supply the outputs of the axes board with 24V. If the fuse is blown, replace it.

2013 'I/O 1 board without voltage'
2014 'I/O 2 board without voltage'
2015 'I/O 3 board without voltage'

DETECTED During execution.

CAUSE The 24V are missing from the outputs of the corresponding I/O board. The fuse might be blown.

SOLUTION Supply the outputs of the corresponding I/O board with 24V. If the fuse is blown, replace it.

2016 'PLC not ready.'

DETECTED During execution.

CAUSE The PLC program is not running. The probable causes might be:
- There is no PLC program
- WATCHDOG error
- The program has been stopped from the monitoring mode.

SOLUTION Restart the PLC program by restarting the PLC.

2017 'CNC RAM memory error'

DETECTED While starting the CNC up or during diagnosis.

CAUSE A RAM memory problem has been detected at the CNC.

SOLUTION Change the CPU board. Contact the Service Department.

2018 'CNC EPROM memory error'

DETECTED While starting the CNC up or during diagnosis..

CAUSE An EPROM memory problem has been detected at the CNC.

SOLUTION Change the EPROM. Contact the Service Department.

2019 'PLC RAM memory error'

DETECTED While starting the CNC up or during diagnosis..

CAUSE A RAM memory problem has been detected at the PLC.

SOLUTION Change the PLC board. Contact the Service Department.

2020 'PLC EPROM memory error'

DETECTED While starting the CNC up or during diagnosis..

CAUSE An EPROM memory problem has been detected at the PLC.

SOLUTION Change the EPROM. Contact the Service Department.

2021 'USER RAM memory error at the CNC. Press any key.'

DETECTED While starting the CNC up or during diagnosis..

CAUSE A user RAM memory problem has been detected at the CNC.

SOLUTION Contact the Service Department.

2022 'CNC system RAM memory error. Press any key.'

DETECTED While starting the CNC up or during diagnosis..

CAUSE A system RAM memory problem has been detected at the CNC.

SOLUTION Contact the Service Department.

2023 'PLC RAM error. Press any key.'

DETECTED While starting the CNC up or during diagnosis..

CAUSE A RAM memory problem has been detected at the PLC.

SOLUTION Contact the Service Department.

2024 'The tracing module has no voltage'

DETECTED During execution.
CAUSE The 24V are missing from the outputs of the tracing board. The fuse might be blown.
SOLUTION Supply the outputs of the tracing board with 24V. If the fuse is blown, replace it.

2025 'Probe feedback error'

DETECTED During execution.
CAUSE The tracing probe is not connected or some cable is connected wrong.
SOLUTION Check the probe connections.

2026 'Maximum probe travel overrun'

DETECTED During execution.
CAUSE The probe has exceeded the maximum deflection allowed by machine parameter.
SOLUTION Reduce the feedrate and check that the probe is not damaged.

2027 'SERCOS chip RAM Error. Press a key.'

DETECTED While starting the CNC up or during diagnosis..
CAUSE A RAM memory problem has been detected at the SERCOS chip.
SOLUTION Change the SERCOS board. Contact the Service Department.

2028 'SERCOS chip version Error. Press a key.'

DETECTED While starting the CNC up.
CAUSE The SERCOS chip version is old.
SOLUTION Change the SERCOS chip. Contact the Service Department.

PLC ERRORS

3000 '(PLC_ERR without description)'

DETECTED During execution.

CAUSE Marks ERR1 through ERR64 have been set to "1".

SOLUTION Check why these marks are set to "1" in the PLC program and act accordingly.

3001 'WATCHDOG in Main Module (PRG).'

DETECTED At any time.

CAUSE/S The probable causes might be:
1.- The main PLC program execution takes longer than the time period set by PLC parameter WAGPRG(P0).
2.- The program is in a loop.

SOLUTION Increase the time period of PLC parameter WAGPRG(P0) or increase the PLC processing speed.
- Insert the CPU TURBO.
- Change the PLC parameter CPUTIME(P26) or general parameter LOOPTIME(P72).

3002 'WATCHDOG in Periodic Module (PE).'

DETECTED At any time.

CAUSE/S The probable causes might be:
1.- The periodic PLC program execution takes longer than the time period set by PLC parameter WAGPER(P1).
2.- The program is in a loop.

SOLUTION Increase the time period of PLC parameter WAGPER(P1) or increase the PLC processing speed.
- Insert the CPU TURBO.
- Change the PLC parameter CPUTIME(P26) or general parameter LOOPTIME(P72).

3003 'Division by zero in PLC.'

DETECTED At any time.

CAUSE The PLC program contains a line whose execution involves a division by zero.

SOLUTION When working with registers, that register may have receive the zero value throughout the program history. Check that the register does not reach the operation with that value.

3004 'PLC Error -> '

DETECTED At any time.

CAUSE An error has been detected on the PLC board.

SOLUTION Change the PLC board. Contact the Service Department.

DRIVE ERRORS

4000 'SERCOS ring error'

DETECTED During execution.

CAUSE SERCOS communication has been interrupted. This could be because there has been an interruption in the connection ring (disconnected or broken fiber link) or the wrong configuration:

1.- The node selector switch position does not match the sercosid.
2.- Parameter P120 (SERSPD) does not match the transmission speed.
3.- The drive version is not compatible with the CNC.
4.- An error has been detected on the SERCOS board.
5.- The transmission speeds are different at the drive and at the CNC.

SOLUTION To check that the connection ring has not been interrupted, verify that the light travels through the optical fiber. If it is due to the wrong configuration, contact the Service Department.

4002 'Drive overload (201)'
4003 'Drive overtemperature (107)'
4004 'Motor overtemperature (108)'
4005 'Heat-sink overtemperature (106)'
4006 'Voltage control error (100...105)'
4007 'Feedback error (600...606)'
4008 'Power bus error (213...215)'
4009 'Overcurrent (212)'
4010 'Power bus overvoltage (304/306)'
4011 'Power bus undervoltage (307)'

DETECTED During execution.

CAUSE An error has been detected at the drive. The number in brackets indicates the standard error number of the drive. Refer to the drive manual for further information.

SOLUTION These types of errors come with messages 4019, 4021, 4022 or 4023 which indicate at which axis drive or spindle drive the error has come up. Refer to the drive manual for the error (number in brackets) and act accordingly.

4016 'Error, undefined class 1'

DETECTED During execution.

CAUSE The drive has detected an error, but it cannot identify it.

SOLUTION Contact the Service Department.

4017 'Drive error'

DETECTED During execution.

CAUSE An error has been detected at the drive which does not match the standard SERCOS errors.

SOLUTION These types of errors come with messages 4019, 4021, 4022 or 4023 which indicate at which axis drive or spindle drive the error has come up. Refer to the drive manual for the error and act accordingly.

4018 'Sercos variable accessing error'

DETECTED During execution.

CAUSE An attempt has been made to read (or write) a SERCOS variable from the CNC, but:
1.- The variable does not exist.
2.- The maximum/minimum values have been exceeded.
3.- The SERCOS variable has variable length
4.- the variable is read-only and cannot be written.

SOLUTION Check that the variable is of the right type for that particular action.

4019 'Axis drive error on: '

DETECTED During execution.

CAUSE These messages come with errors 4002 - 4011. When one of those errors come up, it indicates on which axis it came up.

4021 'Spindle drive error'
4022 '2nd spindle drive error'
4023 'Auxiliary spindle drive error'

DETECTED During execution.

CAUSE These messages come with errors 4002 - 4011. When one of those errors come up, it indicates on which spindle it came up.

4024 'SERCOS error when homing'

DETECTED During execution.

CAUSE The SERCOS home searching command has been executed wrong.

4025 'SERCOS ring error 1'

DETECTED During execution.

CAUSE The time it takes to calculate the axis speed exceeds the cycle time set to transmit to the drive.

SOLUTION Contact the Service Department.

TABLE DATA ERRORS

echk_gen 'CHECKSUM ERROR: GENERAL PARAMETERS Initialize? (ENTER/ESC)'
echk_cab 'CHECKSUM ERROR: SPINDLE PARAMETERS Initialize? (ENTER/ESC)'
echk_cab2 'CHECKSUM ERROR:2nd SPINDLE PARAMETERS Initialize? (ENTER/ESC)'
echk_cax 'CHECKSUM ERROR:AUX.SPINDLE PARAMETERS Initialize? (ENTER/ESC)'
echk_rs1 'CHECKSUM ERROR:SERIAL LINE 1 PARAMETERS Initialize? (ENTER/ESC)'
echk_rs2 'CHECKSUM ERROR:SERIAL LINE 2 PARAMETERS Initialize? (ENTER/ESC)'
echk_plc 'CHECKSUM ERROR:PLC PARAMETERS Initialize? (ENTER/ESC)'

DETECTED While starting the CNC up.
CAUSE Data lost in the tables. Possible RAM error.
SOLUTION By pressing [ENTER] the tables are loaded with default values. If the error persists, contact the Service Department.

echk_org 'CHECKSUM ERROR:ZERO OFFSET TABLE Initialize? (ENTER/ESC)'
echk_psw 'CHECKSUM ERROR:PASSWORD TABLE Initialize? (ENTER/ESC)'

DETECTED While starting the CNC up.
CAUSE Data lost in the tables. Possible RAM error.
SOLUTION By pressing [ENTER] the tables are loaded with default values. If the error persists, contact the Service Department.

echk_ejex 'CHECKSUM ERROR:AXIS X PARAMETERS Initialize? (ENTER/ESC)'
echk_ejey 'CHECKSUM ERROR:AXIS Y PARAMETERS Initialize? (ENTER/ESC)'
echk_ejez 'CHECKSUM ERROR:AXIS Z PARAMETERS Initialize? (ENTER/ESC)'
echk_ejeu 'CHECKSUM ERROR:AXIS U PARAMETERS Initialize? (ENTER/ESC)'
echk_ejev 'CHECKSUM ERROR:AXIS V PARAMETERS Initialize? (ENTER/ESC)'
echk_ejew 'CHECKSUM ERROR:AXIS W PARAMETERS Initialize? (ENTER/ESC)'
echk_ejea 'CHECKSUM ERROR:AXIS A PARAMETERS Initialize? (ENTER/ESC)'
echk_ejeb 'CHECKSUM ERROR:AXIS B PARAMETERS Initialize? (ENTER/ESC)'
echk_ejec 'CHECKSUM ERROR:AXIS C PARAMETERS Initialize? (ENTER/ESC)'

DETECTED While starting the CNC up.
CAUSE Data lost in the axis parameter tables. Possible RAM error.
SOLUTION By pressing [ENTER] the tables are loaded with default values. If the error persists, contact the Service Department.

echk_herr 'CHECKSUM ERROR:TOOL TABLE Initialize? (ENTER/ESC)''
echk_corr 'CHECKSUM ERROR:TOOL OFFSET TABLE Initialize? (ENTER/ESC)'
echk_alm 'CHECKSUM ERROR:MAGAZINE TABLE Initialize? (ENTER/ESC)'
echk_aux 'CHECKSUM ERROR:M FUNCTION TABLE Initialize? (ENTER/ESC)'
echk_husx 'CHECKSUM ERROR:LEADSCREW X TABLE Initialize? (ENTER/ESC)'
echk_husy 'CHECKSUM ERROR:LEADSCREW Y TABLE Initialize? (ENTER/ESC)'
echk_husz 'CHECKSUM ERROR:LEADSCREW Z TABLE Initialize? (ENTER/ESC)'
echk_husu 'CHECKSUM ERROR:LEADSCREW U TABLE Initialize? (ENTER/ESC)'
echk_husv 'CHECKSUM ERROR:LEADSCREW V TABLE Initialize? (ENTER/ESC)'
echk_husw 'CHECKSUM ERROR:LEADSCREW W TABLE Initialize? (ENTER/ESC)'
echk_husa 'CHECKSUM ERROR:LEADSCREW A TABLE Initialize? (ENTER/ESC)'
echk_husb 'CHECKSUM ERROR:LEADSCREW B TABLE Initialize? (ENTER/ESC)'
echk_husc 'CHECKSUM ERROR:LEADSCREW C TABLE Initialize? (ENTER/ESC)'
echk_cru1 'CHECKSUM ERROR:CROSS COMP. TABLE 1 Initialize? (ENTER/ESC)'
echk_cru2 'CHECKSUM ERROR:CROSS COMP. TABLE 2 Initialize? (ENTER/ESC)'
echk_cru3 'CHECKSUM ERROR:CROSS COMP. TABLE 3 Initialize? (ENTER/ESC)'

DETECTED While starting the CNC up.

CAUSE Data lost in the tables. Possible RAM error.

SOLUTION By pressing [ENTER] the tables are loaded with default values. If the error persists, contact the Service Department.

eincx 'Incorrect X axis leadscrew table. Press any key'
eincy 'Incorrect Y axis leadscrew table. Press any key'
eincz 'Incorrect Z axis leadscrew table. Press any key'
eincu 'Incorrect U axis leadscrew table. Press any key'
eincv 'Incorrect V axis leadscrew table. Press any key'
eincw 'Incorrect W axis leadscrew table. Press any key'
einca 'Incorrect A axis leadscrew table. Press any key'
eincb 'Incorrect B axis leadscrew table. Press any key'
eincc 'Incorrect C axis leadscrew table. Press any key'

DETECTED While starting the CNC up.

CAUSE Wrong data in the leadscrew compensation table.

SOLUTION The points must be defined in the table as follows:

- They must be ordered according to their position on the axis starting from the most negative or least positive point to be compensated for.
- The machine reference point must have an error value of zero.
- The error difference between two points cannot be greater than the distance between them.

einx1 'Incorrect cross compensation table 1'
einx2 'Incorrect cross compensation table 2'
einx3 'Incorrect cross compensation table 3'

DETECTED While starting the CNC up.

CAUSE Wrong data in the cross compensation table.

SOLUTION The points must be defined in the table as follows:

- They must be ordered according to their position on the axis starting from the most negative or least positive point to be compensated for.
- The machine reference point must have an error value of zero.

ERRORS IN 8055MC OPERATING MODE

Errors in the surface milling operation.

'SURFACE MILLING: F=0'

DETECTED During execution.
CAUSE Feedrate «F» has the wrong value.
SOLUTION Program a positive feedrate «F» other than zero.

'SURFACE MILLING: S=0'

DETECTED During execution.
CAUSE Spindle speed «S» has the wrong value.
SOLUTION Program a positive «S» other than zero.

'SURFACE MILLING: T=0'

DETECTED During execution.
CAUSE The tool number «T» has not been defined.
SOLUTION The tool number «T» must be other than zero.

'SURFACE MILLING: P=0'

DETECTED During execution.
CAUSE The depth of the SURFACE MILLING «P» has not been defined.
SOLUTION The depth of the SURFACE MILLING «P» must be other than zero.

Errors in the profiling operation 1.

'PROFILING 1: F=0'

DETECTED During execution.
CAUSE Feedrate «F» has the wrong value.
SOLUTION Program a positive feedrate «F» other than zero.

'PROFILING 1: S=0'

DETECTED During execution.
CAUSE Spindle speed «S» has the wrong value.
SOLUTION Program a positive «S» other than zero.

'PROFILING 1: T=0'

DETECTED During execution.
CAUSE The tool number «T» has not been defined.
SOLUTION The tool number «T» must be other than zero.

'PROFILING 1: P=0'

DETECTED During execution.
CAUSE The milling depth «P» has not been defined.
SOLUTION The milling depth «P» must be other than zero.

'PROFILING 1: No profile'

DETECTED During execution.
CAUSE The profile to be machined has not been defined.
SOLUTION The profile must be formed by two points besides the ones for the entry and the exit.

Errors in the profiling operation 2.

'PROFILING 2: T=0'

DETECTED During execution.
CAUSE The tool number «T» has not been defined.
SOLUTION The tool number «T» must be other than zero.

'PROFILING 2: F=0'

DETECTED During execution.
CAUSE Feedrate «F» has the wrong value.
SOLUTION Program a positive feedrate «F» other than zero.

'PROFILING 2: S=0'

DETECTED During execution.
CAUSE Spindle speed «S» has the wrong value.
SOLUTION Program a positive «S» other than zero.

'PROFILING 2: P=0'

DETECTED During execution.
CAUSE The milling depth «P» has not been defined.
SOLUTION The milling depth «P» must be other than zero.

Errors in the pocket profiling operation.

'POCKET PROFILE: F=0'

DETECTED During execution.
CAUSE Feedrate «F» has the wrong value.
SOLUTION Program a positive feedrate «F» other than zero.

'POCKET PROFILE: S=0'

DETECTED During execution.
CAUSE Spindle speed «S» has the wrong value.
SOLUTION Program a positive «S» other than zero.

'POCKET PROFILE: P=0'

DETECTED During execution.
CAUSE The pocket depth «P» has not been defined.
SOLUTION The pocket depth «P» must be other than zero.

'POCKET PROFILE: Wrong penetration angle value'

DETECTED During execution.
CAUSE A penetration angle smaller than 0º or greater than 90º has been programmed.
SOLUTION Program a penetration angle «β» and «Θ» within the 0º to 90º range.

'POCKET PROFILE: Tool diameter smaller than Δ'

DETECTED During execution.
CAUSE The programmed milling step «Δ» is larger than the tool diameter.
SOLUTION Program a milling step «Δ» smaller than the tool diameter or choose a tool with a larger diameter.

'POCKET PROFILE: Finishing tool diameter smaller than δ'

DETECTED During execution.
CAUSE The programmed finishing stock «δ» is greater than the tool diameter.
SOLUTION Program a finishing stock «δ» smaller than the tool diameter or choose a tool with a larger diameter.

Errors in the 3D POCKET PROFILE operation.

'3D POCKET PROFILE: F=0'

DETECTED During execution.
CAUSE Feedrate «F» has the wrong value.
SOLUTION Program a positive feedrate «F» other than zero.

'3D POCKET PROFILE: S=0'

DETECTED During execution.
CAUSE Spindle speed «S» has the wrong value.
SOLUTION Program a positive «S» other than zero.

'3D POCKET PROFILE: P=0'

DETECTED During execution.
CAUSE The pocket depth «P» has not been defined.
SOLUTION The pocket depth «P» must be other than zero.

'3D POCKET PROFILE: Wrong penetration angle value'

DETECTED During execution.
CAUSE A penetration angle smaller than 0º or greater than 90º has been programmed
SOLUTION Program a penetration angle «β» and «Θ» within the 0º to 90º range.

'3D POCKET PROFILE: Tool diameter smaller than Δ'

DETECTED During execution.
CAUSE The programmed milling step «Δ» is larger than the tool diameter.
SOLUTION Program a milling step «Δ» smaller than the tool diameter or choose a tool with a larger diameter.

'3D POCKET PROFILE: Finishing tool smaller than δ'

DETECTED During execution.
CAUSE The programmed finishing stock «δ» is greater than the tool diameter.
SOLUTION Program a finishing stock «δ» smaller than the tool diameter or choose a tool with a larger diameter.

Errors in the rectangular pocket operation 1.

'RECTANGULAR POCKET 1: F=0'

DETECTED During execution.
CAUSE Feedrate «F» has the wrong value.
SOLUTION Program a positive feedrate «F» other than zero.

'RECTANGULAR POCKET 1: S=0'

DETECTED During execution.
CAUSE Spindle speed «S» has the wrong value.
SOLUTION Program a positive «S» other than zero.

'RECTANGULAR POCKET 1: T=0'

DETECTED During execution.
CAUSE The tool number «T» has not been defined.
SOLUTION The tool number «T» must be other than zero.

'RECTANGULAR POCKET 1: P=0'

DETECTED During execution.
CAUSE The pocket depth «P» has not been defined.
SOLUTION The pocket depth «P» must be other than zero.

'RECTANGULAR POCKET 1: Tool diameter smaller than Δ'

DETECTED During execution.
CAUSE The programmed milling step «Δ» is larger than the tool diameter.
SOLUTION Program a milling step «Δ» smaller than the tool diameter or choose a tool with a larger diameter.

'RECTANGULAR POCKET 1: Tool diameter larger than the pocket'

DETECTED During execution.
CAUSE The tool diameter is greater than one of the «H» or «L» dimensions of the pocket.
SOLUTION Choose a tool with a smaller diameter to machine the pocket.

'RECTANGULAR POCKET 1: Finishing tool diameter smaller than δ'

DETECTED During execution.
CAUSE The programmed finishing stock «δ» is greater than the tool diameter.
SOLUTION Program a finishing stock «δ» smaller than the tool diameter or choose a tool with a larger diameter.

Errors in the rectangular pocket operation 2.

'RECTANGULAR POCKET 2: F=0'

DETECTED During execution.
CAUSE Feedrate «F» has the wrong value.
SOLUTION Program a positive feedrate «F» other than zero.

'RECTANGULAR POCKET 2: S=0'

DETECTED During execution.
CAUSE Spindle speed «S» has the wrong value.
SOLUTION Program a positive «S» other than zero.

'RECTANGULAR POCKET 2: P=0'

DETECTED During execution.
CAUSE The pocket depth «P» has not been defined.
SOLUTION The pocket depth «P» must be other than zero.

'RECTANGULAR POCKET 2: Wrong penetration angle value'

DETECTED During execution.
CAUSE A penetration angle smaller than 0º or greater than 90º has been programmed
SOLUTION Program a penetration angle «β» and «Θ» within the 0º to 90º range.

'RECTANGULAR POCKET 2: Tool diameter smaller than Δ'

DETECTED During execution.
CAUSE The programmed milling step «Δ» is larger than the tool diameter.
SOLUTION Program a milling step «Δ» smaller than the tool diameter or choose a tool with a larger diameter.

'RECTANGULAR POCKET 2: Tool diameter larger than the pocket'

DETECTED During execution.
CAUSE The tool diameter is greater than one of the «H» or «L» dimensions of the pocket.
SOLUTION Choose a tool with a smaller diameter to machine the pocket.

'RECTANGULAR POCKET 2: Finishing tool diameter smaller than δ'

DETECTED During execution.
CAUSE The programmed finishing stock «δ» is greater than the tool diameter.
SOLUTION Program a finishing stock «δ» smaller than the tool diameter or choose a tool with a larger diameter.

Errors in the circular pocket operation.

'CIRCULAR POCKET: F=0'

DETECTED During execution.
CAUSE Feedrate «F» has the wrong value.
SOLUTION Program a positive feedrate «F» other than zero.

'CIRCULAR POCKET: S=0'

DETECTED During execution.
CAUSE Spindle speed «S» has the wrong value.
SOLUTION Program a positive «S» other than zero.

'CIRCULAR POCKET: P=0'

DETECTED During execution.
CAUSE The pocket depth «P» has not been defined.
SOLUTION The pocket depth «P» must be other than zero.

'CIRCULAR POCKET: Wrong penetration angle value'

DETECTED During execution.
CAUSE A penetration angle smaller than 0º or greater than 90º has been programmed
SOLUTION Program a penetration angle «β» and «Θ» withiν the 0º to 90º range.

'CIRCULAR POCKET: Tool diameter smaller than Δ'

DETECTED During execution.
CAUSE The programmed milling step «Δ» is larger than the tool diameter.
SOLUTION Program a milling step «Δ» smaller than the tool diameter or choose a tool with a larger diameter.

'CIRCULAR POCKET: Tool diameter larger than the pocket'

DETECTED During execution.
CAUSE The tool radius is larger than the pocket radius «R».
SOLUTION Choose a tool with a smaller diameter to machine the pocket.

'CIRCULAR POCKET: Finishing tool smaller than δ'

DETECTED During execution.
CAUSE The programmed finishing stock «δ» is greater than the tool diameter.
SOLUTION Program a finishing stock «δ» smaller than the tool diameter or choose a tool with a larger diameter.

Errors in the rectangular boss milling operation.

'RECTANGULAR BOSS: F=0'

DETECTED During execution.
CAUSE Feedrate «F» has the wrong value.
SOLUTION Program a positive feedrate «F» other than zero.

'RECTANGULAR BOSS: S=0'

DETECTED During execution.
CAUSE Spindle speed «S» has the wrong value.
SOLUTION Program a positive «S» other than zero.

'RECTANGULAR BOSS: P=0'

DETECTED During execution.
CAUSE The height of the boss «P» has not been defined.
SOLUTION The height of the boss «P» must be other than zero.

'RECTANGULAR BOSS: Tool diameter smaller than Δ'

DETECTED During execution.
CAUSE The programmed milling step «Δ» is larger than the tool diameter.
SOLUTION Program a milling step «Δ» smaller than the tool diameter or choose a tool with a larger diameter.

'RECTANGULAR BOSS: Finishing tool diameter smaller than δ'

DETECTED During execution.
CAUSE The programmed finishing stock «δ» is greater than the tool diameter.
SOLUTION Program a finishing stock «δ» smaller than the tool diameter or choose a tool with a larger diameter.

Errors in the circular boss milling operation.

'CIRCULAR BOSS: F=0'

DETECTED During execution.
CAUSE Feedrate «F» has the wrong value.
SOLUTION Program a positive feedrate «F» other than zero.

'CIRCULAR BOSS: S=0'

DETECTED During execution.
CAUSE Spindle speed «S» has the wrong value.
SOLUTION Program a positive «S» other than zero.

'CIRCULAR BOSS: P=0'

DETECTED During execution.
CAUSE The height of the boss «P» has not been defined.
SOLUTION The height of the boss «P» must be other than zero.

'CIRCULAR BOSS: Tool diameter smaller than Δ'

DETECTED During execution.
CAUSE The programmed milling step «Δ» is larger than the tool diameter.
SOLUTION Program a milling step «Δ» smaller than the tool diameter or choose a tool with a larger diameter.

'CIRCULAR BOSS: Finishing tool smaller than δ'

DETECTED During execution.
CAUSE The programmed finishing stock «δ» is greater than the tool diameter.
SOLUTION Program a finishing stock «δ» smaller than the tool diameter or choose a tool with a larger diameter.

Errors in the center punching operation.

'CENTER PUNCHING: F=0'

DETECTED During execution.
CAUSE Feedrate «F» has the wrong value.
SOLUTION Program a positive feedrate «F» other than zero.

'CENTER PUNCHING: S=0'

DETECTED During execution.
CAUSE Spindle speed «S» has the wrong value.
SOLUTION Program a positive «S» other than zero.

'CENTER PUNCHING: T=0'

DETECTED During execution.
CAUSE The tool number «T» has not been defined.
SOLUTION The tool number «T» must be other than zero.

'CENTER PUNCHING: P=0'

DETECTED During execution.
CAUSE The depth of the CENTER PUNCHING operation «P» has not been defined.
SOLUTION The depth of the CENTER PUNCHING operation «P» must be other than zero.

'CENTER PUNCHING: ø=0'

DETECTED During execution.
CAUSE The point diameter «ø» has not been defined.
SOLUTION The point diameter «ø» must be positive and other than zero.

'CENTER PUNCHING: α=0'

DETECTED During execution.
CAUSE The angle of the punch tip «α» has not been defined.
SOLUTION The angle of the punch tip «α» must be positive and other than zero.

Errors in the drilling operation 1.

'DRILLING 1: F=0'

DETECTED During execution.
CAUSE Feedrate «F» has the wrong value.
SOLUTION Program a positive feedrate «F» other than zero.

'DRILLING 1: S=0'

DETECTED During execution.
CAUSE Spindle speed «S» has the wrong value.
SOLUTION Program a positive «S» other than zero.

'DRILLING 1: T=0'

DETECTED During execution.
CAUSE The tool number «T» has not been defined.
SOLUTION The tool number «T» must be other than zero.

'DRILLING 1: P=0'

DETECTED During execution.
CAUSE The DRILLING depth «P» has not been defined.
SOLUTION The DRILLING depth «P» must be other than zero.

Errors in the drilling operation 2

'DRILLING 2: F=0'

DETECTED During execution.
CAUSE Feedrate «F» has the wrong value.
SOLUTION Program a positive feedrate «F» other than zero.

'DRILLING 2: S=0'

DETECTED During execution.
CAUSE Spindle speed «S» has the wrong value.
SOLUTION Program a positive «S» other than zero.

'DRILLING 2: T=0'

DETECTED During execution.
CAUSE The tool number «T» has not been defined.
SOLUTION The tool number «T» must be other than zero.

'DRILLING 2: P=0'

DETECTED During execution.
CAUSE The DRILLING depth «P» has not been defined.
SOLUTION The DRILLING depth «P» must be other than zero.

'DRILLING 2: B=0'

DETECTED During execution.
CAUSE The withdrawal distance «B» after each drilling peck has not been defined.
SOLUTION The withdrawal distance «B» after each drilling peck must be other than zero.

Errors in the tapping operation.

'TAPPING: F=0'

DETECTED During execution.
CAUSE Feedrate «F» has the wrong value.
SOLUTION Program a positive feedrate «F» other than zero.

'TAPPING: S=0'

DETECTED During execution.
CAUSE Spindle speed «F» has the wrong value.
SOLUTION Program a positive «F» other than zero.

'TAPPING: T=0'

DETECTED During execution.
CAUSE The tool number «T» has not been defined.
SOLUTION The tool number «T» must be other than zero.

'TAPPING: P=0'

DETECTED During execution.
CAUSE The TAPPING depth «P» has not been defined.
SOLUTION The TAPPING depth «P» must be other than zero.

Errors in the reaming operation.

'REAMING: F=0'

DETECTED During execution.
CAUSE Feedrate «F» has the wrong value.
SOLUTION Program a positive feedrate «F» other than zero.

'REAMING: S=0'

DETECTED During execution.
CAUSE Spindle speed «S» has the wrong value.
SOLUTION Program a positive «S» other than zero.

'REAMING: T=0'

DETECTED During execution.
CAUSE The tool number «T» has not been defined.
SOLUTION The tool number «T» must be other than zero.

'REAMING: P=0'

DETECTED During execution.
CAUSE The REAMING depth «P» has not been defined.
SOLUTION The REAMING depth «P» must be other than zero.

Errors in the boring operation.

'BORING: F=0'

DETECTED During execution.
CAUSE Feedrate «F» has the wrong value.
SOLUTION Program a positive feedrate «F» other than zero.

'BORING: S=0'

DETECTED During execution.
CAUSE Spindle speed «S» has the wrong value.
SOLUTION Program a positive «S» other than zero.

'BORING: T=0'

DETECTED During execution.
CAUSE The tool number «T» has not been defined.
SOLUTION The tool number «T» must be other than zero.

'BORING: P=0'

DETECTED During execution.
CAUSE The BORING depth «P» has not been defined.
SOLUTION The BORING depth «P» must be other than zero.

Errors in the positioning operations.

'LINEAR POSITIONING: Wrong I value'

DETECTED During execution.
CAUSE The distance between points «I» has the wrong value and it does not allow machining an entire number of points.
SOLUTION Check the data entered.

'CIRCULAR POSITIONING: Wrong ß value'

DETECTED During execution.
CAUSE The angular distance between points «β» has the wrong value and it does not allow machining an entire number of points.
SOLUTION Check the data entered.

'RECTANGULAR POSITIONING: Wrong Ix/Iy value'

DETECTED During execution.
CAUSE One of the distances between points «Ix/Iy» has the wrong value and it does not allow machining an entire number of points.
SOLUTION Check the data entered.

'GRID PATTERN POSITIONING: Wrong Ix/Iy value'

DETECTED During execution.
CAUSE One of the distances between points «Ix/Iy» has the wrong value and it does not allow machining an entire number of points.
SOLUTION Check the data entered.

NOTES

NOTES

ALPHABETICAL INDEX

'* axis cannot be synchronized' 49
'* axis feedback error' 52
'* axis following error limit overrun' 50
'* axis hard limit overrun' 50
'* axis locked' 49
'* axis maximum feed exceeded' 49
'* axis range exceeded' 48
'* axis servo error' 50
'* axis soft limit overrun' 49
'* axis work zone 1 overrun' 49
'* axis work zone 2 overrun' 49
'* axis work zone 3 overrun' 50
'* axis work zone 4 overrun' 51
'2nd spindle drive error' 57
'3D POCKET PROFILE: F=0' 62
'3D POCKET PROFILE: Finishing tool smaller than δ' 63
'3D POCKET PROFILE: P=0' 62
'3D POCKET PROFILE: S=0' 62
'3D POCKET PROFILE: Tool diameter smaller than Δ' 63
'3D POCKET PROFILE: Wrong penetration angle value' 63

A

'A parameter required by the canned cycle has not been programmed' 39
'A step greater than the tool diameter has been programmed' 37
'A subroutine is not allowed for automatic range change' 38
'A tool cannot be programmed with G48 active' 32
'A tool cannot be programmed with G48 active' 37
'A tool change has been programmed without M06' 36
'A tool with no radius has been programmed' 37
'Access to a variable with non-permitted index' 41
'Analog inputs: ANAI(1-8) = +/-5 Volts.' 27
'Analog output not available.' 15
'Analog outputs: ANAO(1-8) = +/-10 Volts.' 27
'Angle coordinate programmed incorrectly' 44
'Arc programmed with radius too small or complete circle' 44
'ASIN/ACOS range exceeded.' 18
'Auxiliary spindle drive error' 57
'Axes board without voltage' 52
'Axes X, Y and Z must exist.' 29
'Axes X, Y or Z slaved or synchronized.' 29
'Axis does not exist.' 11
'Axis drive error on: ' 57

B

'Base zero with positive exponent.' 17
'Beginning of compensation without a straight path' 43
'Block cannot be executed while running another program' 20
'Block incompatible when defining a profile.' 5
'Block not allowed in MDI or during tool inspection' 45
'BORING: F=0' 67
'BORING: P=0' 68
'BORING: S=0' 68
'BORING: T=0' 68

C

'Canned cycle does not exist' 39
'CENTER PUNCHING: **α=0' 66
'CENTER PUNCHING: F=0' 66
'CENTER PUNCHING: ø=0' 66
'CENTER PUNCHING: P=0' 66
'CENTER PUNCHING: S=0' 66
'CENTER PUNCHING: T=0' 66
'Chamfer programmed incorrectly' 34
'Chamfer value too large' 34
'CHECKSUM ERROR: GENERAL PARAMETERS ' 58
'CHECKSUM ERROR: SPINDLE PARAMETERS ' 58
'CHECKSUM ERROR:2nd SPINDLE PARAMETERS ' 58
'CHECKSUM ERROR:AUX.SPINDLE PARAMETERS ' 58
'CHECKSUM ERROR:AXIS * PARAMETERS ' 58
'CHECKSUM ERROR:CROSS COMP. TABLE 1 ' 59
'CHECKSUM ERROR:CROSS COMP. TABLE 2 ' 59
'CHECKSUM ERROR:CROSS COMP. TABLE 3 ' 59
'CHECKSUM ERROR:LEADSCREW * TABLE ' 59
'CHECKSUM ERROR:M FUNCTION TABLE ' 59
'CHECKSUM ERROR:MAGAZINE TABLE ' 59
'CHECKSUM ERROR:PASSWORD TABLE ' 58
'CHECKSUM ERROR:PLC PARAMETERS ' 58
'CHECKSUM ERROR:SERIAL LINE 1 PARAMETERS ' 58
'CHECKSUM ERROR:SERIAL LINE 2 PARAMETERS ' 58
'CHECKSUM ERROR:TOOL OFFSET TABLE ' 59
'CHECKSUM ERROR:TOOL TABLE '' 59
'CHECKSUM ERROR:ZERO OFFSET TABLE ' 58
'Circle with zero radius' 46
'Circular (helical) interpolation not possible.' 27
'CIRCULAR BOSS: F=0' 65
'CIRCULAR BOSS: Finishing tool smaller than δ' 65
'CIRCULAR BOSS: P=0' 65
'CIRCULAR BOSS: S=0' 65
'CIRCULAR BOSS: Tool diameter smaller than Δ' 65
'Circular path programmed incorrectly' 46
'CIRCULAR POCKET: F=0' 64
'CIRCULAR POCKET: Finishing tool smaller than δ' 65
'CIRCULAR POCKET: P=0' 64
'CIRCULAR POCKET: S=0' 64
'CIRCULAR POCKET: Tool diameter larger than the pocket' 64
'CIRCULAR POCKET: Tool diameter smaller than Δ' 64
'CIRCULAR POCKET: Wrong penetration angle value' 64
'CIRCULAR POSITIONING: Wrong ß value' 68
'CNC EPROM memory error' 53
'CNC RAM memory error' 53
'CNC system RAM memory error. Press any key.' 53
'Compensation plane change' 44
'Compensation radius too large' 43
'Complete Table.' 30
'Coupled * axis following error difference too large' 50
'Cycle does not exist.' 28

D

'Deflection out of range.' 29
'Division by zero in PLC.' 55
'Division by zero.' 17
'Do not modify the active tool or the next one.' 22
'Do not program «Q» with parameter M19TYPE=0.' 33
'Do not program a GANTRY axis.' 13
'Do not program a slaved axis.' 12
'Do not program a slaved axis' 48
'Do not program a zero offset without cancelling the previous one.' 31
'Do not program formats greater than 6.5 .' 19
'Do not program labels by parameters.' 3
'Do not program tracing axes.' 29
'Do not switch axes already switched over' 37
'Do not switch axes over or back while G15, G23, G48 or G49 are active' 37
'Do not use high level to change active tool or next one' 36
'Don't program G33 ,G95 or M19 S with no spindle encoder' . 25
'DRILLING 1: F=0' 66

'DRILLING 1: P=0' 66
'DRILLING 1: S=0' 66
'DRILLING 1: T=0' 66
'DRILLING 2: B=0' 67
'DRILLING 2: F=0' 66
'DRILLING 2: P=0' 67
'DRILLING 2: S=0' 66
'DRILLING 2: T=0' 67
'Drive error' 56
'Drive overload (201)' 56
'Drive overtemperature (107)' 56

E

'ELSE not associated with IF.' 13
'Empty line.' 1
'End of compensation without a straight path' 43
'Error when programming drilling an irregular pocket' 40
'Error, undefined class 1' 56
'Expecting "(".' 17
'Expecting ")".' 16
'Expecting ",".' 17
'Expecting "=".' 16
'Expecting a message.' 14
'Expecting a parameter' 15
'External emergency activated' 52

F

'Feedback error (600...606)' 56
'First point programmed wrong when selecting profile' 32
'For G28 or G29, a second spindle is required.' 31
'Format +/- 5.5.' 24
'Function not possible from PLC.' 28

G

'G2 or G3 not allowed when programming a canned cycle.' 6
'G23 has not been programmed.' 29
'G43 cannot be programmed with G48 active' 35
'G48 cannot be programmed with G43 active' 35
'G49 T X Y Z S, X Y Z A B C , or, X Y Z Q R S.' 6
'G51 [A] E' 21
'G60: [A] /X I K/(2) [P Q R S T U V].' 6
'G61-2: [A B] /X I J/(2) Y J D (2)/ [P Q R S T U V].' 7
'G63: X Y /I K/(1) [C P][P Q R S T U V].' 7
'G64: X Y /I K/(1) [C P][P Q R S T U V.' 7
'G65: X Y /A I/(1) [C P].' 8
'G66 must be programmed before G67 and G68.' 26
'G66: [D H][R I][C J][F K] S E [Q].' 8
'G67. Tool radius too large' 37
'G67: [A] B [C] [I] [R] [K] [V].' 8
'G68. Tool radius too large' 37
'G68: [B] [L] [Q] [J] [I] [R] [K].' 9
'G69: I B [C D H J K L R].' 9
'G79 not allowed when there is no active canned cycle.' 25
'G8 defined incorrectly' 35
'G81-84-85-86-89: I [K].' 9
'G82: I K.' 9
'G83: I J.' 10
'G87: I J K B [C] [D] [H] [L] [V].' 10
'G88: I J B [C] [D] [H] [L] [V].' 10
'G96 only possible with analog spindle.' 27
'GRID PATTERN POSITIONING: Wrong Ix/Iy value' 68

H

'Heat-sink overtemperature (106)' 56
'Helical path programmed incorrectly' 46
'High level blocks not allowed when defining a profile.' 5
'HIRTH axis: program only integer values.' 13

I

'I/O 1 board without voltage' 52
'I/O 2 board without voltage' 52
'I/O 3 board without voltage' 52
'Improper data format' 2
'Improper data order.' 1
'Improper data' 1
'Inch programming limit exceeded.' 25
'Incompatible G functions.' 2
'Incomplete Coordinates.' 10
'Incomplete operation.' 16
'Incorrect * axis feedrate parameter' 49
'Incorrect * axis leadscrew table. Press any key' 59
'Incorrect access to PLC variables' 41
'Incorrect active plane and longitudinal axis.' 29
'Incorrect axis.' 21
'Incorrect Coordinates.' 11
'Incorrect cross compensation table 1' 59
'Incorrect cross compensation table 2' 59
'Incorrect cross compensation table 3' 59
'Incorrect cross compensation table parameters' 60
'Incorrect digitizing method.' 30
'Incorrect expression.' 16
'Incorrect message.' 24
'Incorrect number of bits.' 25
'Incorrect operation.' 16
'Incorrect order of axes.' 11
'Incorrect parametric programming.' 25
'Incorrect range change' 38
'Incorrect tracing method.' 30
'Incorrect variable value' 41
'Insufficient accelerations for the programmed threadcutting feedrate' 47
'Insufficient memory.' 25
'Invalid G function after first point of profile' 32
'Invalid G function when selecting a profile' 31
'Invalid parameter value in canned cycle' 40
'Invalid programming after first point of profile' 32

J

'Jog movement out of limits' 35
'Jump to an undefined label' 42

L

'Label cannot be searched' 43
'Label not defined' 42
'Leadscrew: Position-Error.' 21
'LINEAR POSITIONING: Wrong I value' 68
'Local parameters not accessible' 41
'Local parameters not accessible' 42
'Local parameters not allowed.' 20
'Logarithm of zero or negative number.' 17

M

'M function: M4 S4 bits(8).' 21
'Magazine is not RANDOM.' 22
'Magazine: P(1-255) = T(1-9999).' 21
'Maximum probe travel overrun' 54
'Maximum temperature exceeded' 52
'Modal subroutines cannot be programmed.' 27
'Motor overtemperature (108)' 56

N

'Negative base with decimal exponent.' 18
'Negative radius in polar coordinates' 44
'Nesting exceeded.' 42
'Next tool only possible in machining centers.' 22
'No compensation is permitted.' 31
'No more G functions allowed in the block' 3

'No more information allowed in the block.' 2
'No more M functions allowed in the block' 3
'No negative radius allowed with absolute coordinates' 26
'Nonexistent G function' 3
'Nonparametric assignment after first point of profile' 32
'Not enough information about the path' 34
'Not enough room for the automatic range change M code' 38
'Number of repetitions not possible.' 3
'Numerical format exceeded.' 24

O

'Offset D0 does not exist.' 22
'Offset: D3 R L I K.' 20
'Only one HIRTH axis per block is allowed.' 26
'OPEN is missing.' 14
'Option not available.' 28
'Overcurrent (212)' 56

P

'Parameter does not exist.' 15
'Parameter range protected. Cannot be written. P297, P298' 51
'Part surface coordinate not programmed in irregular pocket' ... 41
'Password: use uppercase/lowercase letters or digits.' 26
'Pitch programmed incorrectly.' 12
'Plane change during rounding or chamfering' 34
'Plane change during tool inspection' 45
'Plane profile open in irregular pocket' 41
'(PLC_ERR without description)' 55
'PLC EPROM memory error' 53
'PLC Error -> ' 55
'PLC not ready.' 53
'PLC RAM error. Press any key.' 53
'PLC RAM memory error' 53
'POCKET PROFILE: F=0' 62
'POCKET PROFILE: Finishing tool diameter smaller than ε' 62
'POCKET PROFILE: P=0' 62
'POCKET PROFILE: S=0' 62
'POCKET PROFILE: Tool diameter smaller than Δ' 62
'POCKET PROFILE: Wrong penetration angle value' 62
'Point incompatible with active plane.' 12
'Point within the forbidden zone 1' 47
'Point within the forbidden zone 2' 47
'Point within the forbidden zone 3' 47
'Point within the forbidden zone 4' 47
'Polar coordinates not allowed.' 11
'Position-only rotary axis: Absolute values 0 - 359.9999' 26
'Power bus error (213...215)' 56
'Power bus overvoltage (304/306)' 56
'Power bus undervoltage (307)' 56
'Preset of rotary axes: Values between 0-359.9999. ' 30
'Probe axes out of alignment.' 47
'Probe feedback error' 54
'Probe signal has not been received' 45
'PROFILING 1: F=0' 61
'PROFILING 1: No profile' 61
'PROFILING 1: P=0' 61
'PROFILING 1: S=0' 61
'PROFILING 1: T=0' 61
'PROFILING 2: F=0' 62
'PROFILING 2: P=0' 62
'PROFILING 2: S=0' 62
'PROFILING 2: T=0' 61
'Program columns 0 thru 79.' 19
'Program A (append) or D (delete).' 28
'Program A from 0 to 255' 31
'Program already exists.' 14
'Program another softkey.' 18
'Program another window.' 18
'Program axes.' 11
'Program cannot be opened.' 43
'Program channel 0(CNC),1(PLC) or 2(DNC).' 15
'Program column number.' 18
'Program DNC1/2, HD or CARD A (optional).' 28
'Program does not exist.' 14
'Program error number 0 thru 9999.' 16
'Program F, S, T, D before the M functions.' 3
'Program G27 only when tracing a profile.' 30
'Program G36-G39 with R+5.5.' 4
'Program INPUT.' 19
'Program inputs 0 thru 25.' 19
'Program label N(0-9999).' 13
'Program maximum X' 33
'Program maximum Y' 33
'Program maximum Z' 33
'Program minimum Y' 33
'Program minimum Z' 33
'Program nesting not allowed.' 31
'Program numerical format.' 19
'Program P3 = value.' 21
'Program pages 0 thru 255.' 19
'Program pitch.' 12
'Program Q between +/-359.9999.' 32
'Program row number.' 18
'Program rows 0 thru 20.' 19
'Program softkeys 1 thru 7.' 18
'Program subroutine number 1 thru 9999.' 13
'Program windows 0 thru 25.' 19
'Program: G15 axis.' 4
'Program: G16 axis-axis.' 4
'Program: G22 K(1/2/3/4) S(0/1/2).' 4
'Program: G52 axis +/-5.5.' 30
'Program: G72 S5.5 or axes.' 5
'Program: G73 Q (angle) I J (center).' 5
'Program: G77 axes (2 thru 6).' 6
'Program: G93 I J.' 6
'Program: work zone K1, K2, K3 or K4.' 4
'Programming not allowed while in tracing mode.' 28
'Programming not permitted while G47-G49 are active.' 31

R

'Radius comp. not possible when positioning rotary axis' 44
'Range exceeded' 45
'Read-only variable.' 15
'REAMING: F=0' 67
'REAMING: P=0' 67
'REAMING: S=0' 67
'REAMING: T=0' 67
'RECTANGULAR BOSS: F=0' 65
'RECTANGULAR BOSS: Finishing tool diameter smaller than δ' 65
'RECTANGULAR BOSS: P=0' 65
'RECTANGULAR BOSS: S=0' 65
'RECTANGULAR BOSS: Tool diameter smaller than Δ' 65
'RECTANGULAR POCKET 1: F=0' 63
'RECTANGULAR POCKET 1: Finishing tool diameter smaller than δ' 63
'RECTANGULAR POCKET 1: P=0' 63
'RECTANGULAR POCKET 1: S=0' 63
'RECTANGULAR POCKET 1: T=0' 63
'RECTANGULAR POCKET 1: Tool diameter larger than the pocket' 63
'RECTANGULAR POCKET 1: Tool diameter smaller than Δ' ... 63
'RECTANGULAR POCKET 2: F=0' 63
'RECTANGULAR POCKET 2: Finishing tool diameter smaller than δ' 64
'RECTANGULAR POCKET 2: P=0' 64
'RECTANGULAR POCKET 2: S=0' 64
'RECTANGULAR POCKET 2: Tool diameter larger than the pocket' 64
'RECTANGULAR POCKET 2: Tool diameter smaller than Δ' .. 64
'RECTANGULAR POCKET 2: Wrong penetration angle value' 64
'RECTANGULAR POSITIONING: Wrong Ix/Iy value' 68
'Repeated information' 2
'Repeated subroutine.' 14
'Repositioning not allowed.' 29

'RET not associated to a subroutine' 42
'Rotary axis: Absolute values (G90) within +/-359.9999.' 27
'Rounding in last block' 34
'Rounding radius too large ' 34

S

'S has been programmed without an active range' 38
'S not programmed in G95 or threadcutting' 39
'S programmed too large' 38
'Self-intersecting plane-profile in irregular pocket' 40
'SERCOS chip RAM Error. Press a key.' 54
'SERCOS chip version Error. Press a key.' 54
'SERCOS error when homing' 57
'SERCOS ring error 1' 57
'SERCOS ring error' 56
'Sercos variable accessing error' 57
'Spindle drive error' 57
'Spindle feedback error' 52
'Spindle following error limit overrun' 48
'Spindle locked' 48
'Spindle speed range not defined for M19' 38
'Spindle travel limit overrun' 48
'Square root of a negative number.' 17
'Step in a straight path' 44
'Step in circular path' 44
'Subroutine not available in program' 43
'Subroutine not defined' 42
'SURFACE MILLING: F=0' 61
'SURFACE MILLING: P=0' 61
'SURFACE MILLING: S=0' 61
'SURFACE MILLING: T=0' 61

T

'Table limits exceeded.' 20
'Tangential exit programmed incorrectly' 34
'TAPPING: F=0' 67
'TAPPING: P=0' 67
'TAPPING: S=0' 67
'TAPPING: T=0' 67
'Text too long.' 24
'The axis cannot be programmed after first point of profile' 32
'The canned cycle is missing a tool offset' 36
'The main program cannot have a subroutine.' 14
'The position of a special tool is set.' 22
'The program cannot be executed.' 43
'The program is not accessible' 27
'The programmed axis is not longitudinal.' 26
'The Spindle cannot be referenced (homed)' 46
'The tool is not in the tool magazine' 36
'The tracing module has no voltage' 54
'The window must be previously defined.' 27
'There is no empty pocket in the tool magazine' 36
'There is no information for arctangent in irregular pocket' 35
'There is no information on previous path' 35
'There is no subroutine associated with G74' 45
'There is no tool of the same family to replace it' 36
'This command can only be executed in the user channel.' 20
'This G or M function must be alone.' 3
'Tool not defined in tool table' 36
'Tool not defined.' 22
'Tool offset does not exist' 28
'Tool T must be programmed with G67 and G68.' 25
'Tool T0 does not exist.' 22
'Tool: T4 D3 F3 N5 R5(.2).' 20

U

'User channel: Do not program geometric aides, comp. or cycles' 20
'USER RAM memory error at the CNC. Press any key.' 53

V

'Values 0 thru 100.' 23
'Values 0 thru 2.' 23
'Values 0 thru 255.' 23
'Values 0 thru 3.' 23
'Values 0 thru 32767.' 24
'Values 0 thru 4.' 23
'Values 0 thru 6.' 30
'Values 0 thru 65535.' 24
'Values 0 thru 9.' 23
'Values 0 thru 9999.' 24
'Voltage control error (100...105)' 56

W

'WATCHDOG in Main Module (PRG).' 55
'WATCHDOG in Periodic Module (PE).' 55
'WBUF can only be executed in user channel while editing' 20
'Work zone limit range exceeded' 47
'Write +/-.' 23
'Write 0/1.' 22
'Write ON/OFF.' 23
'Write YES/NO.' 23
'Wrong depth-profile in irregular pocket' 40
'Wrong graphic limits' 33
'Wrong password.' 26
'Wrong plane in tangential path' 35
'Wrong profile intersection in irregular pocket with islands' 50
'Wrong reference plane coordinate in canned cycle' 41
'Wrong sercosid parameters for axes and spindle' 60
'Wrong tool position prior to canned cycle' 40
'Wrong work zone boundaries' 48

Z

'Zero offset range exceeded' 46
'Zero offset: G54-59 axes (1-5).' 21